66th Annual Meeting of the APS Division of Fluid Dynamics

Pittsburgh, Pennsylvania

http://www.aps.org/meetings/meeting.cfm?name=DFD13

Sunday, November 24, 2013 8:00AM - 9:57AM –

Session A1 Geophysical: Atmospheric I 323 - William Layton, University of Pittsburgh

8:00AM A1.00001 Coupling building-resolving LES with meso-scale NWP: effect of the simulation parameters KYONGMIN YEO, IBM TJ Watson Research Center — The effects of the simulation parameters on the scalar dispersion in an urban area are reported. To study scalar dispersion under realistic meteorological conditions, the building-resolving large-eddy simulations (LES) are driven by downscaling boundary conditions from a numerical weather prediction model (NWP). Here, we focus on the changes in the dispersion characteristics under the different downscaling modes and computational parameters. Two sets of numerical simulations are performed for transient morning and unstable day-time atmospheric boundary layers. It is shown that the scalar dispersion is strongly affected by the downscaling method. The computational domain size also has a significant effect on the scalar dispersion even for the ground release events.

8:13AM A1.00002 Measurements of Roughness Length and Displacement Heights in Model Urban Canopies , AUVI RAHMAN, PABLO HUQ, University of Delaware, FERNANDO CAMELLI, George Mason University, UD-GMU COLLABORATION — We present the results of roughness length and displacement height based on PIV velocity measurements in a water tunnel experiment of flow over idealized models of urban canopies. Experiments were conducted with large roughness elements of regular arrays of buildings of uniform height with aspect ratios of 1 and 3. Mean velocity profile above the canopy is described by the log law and a simple optimization procedure to compute the roughness length and displacement height has been developed. Laterally averaged values of displacement height d/H increase from 0 to 1 with plan area density λ_{p} of the urban canopy. In contrast, laterally averaged roughness height z_0/H increases to a maximum value (as λ_{f} approaches a value of 0.2) and then decreases to zero. We present data for effective roughness heights (z_0+d)/H as a function of aspect ratio H/w_{b} of buildings. This also reveals three categories of values: street canyon, building wake, and laterally averaged values. Measurements taken at the centerline of canyons form a lower bound on the effective roughness height whereas measurements behind building wakes form an upper bound. Laterally averaged values of friction velocity u_*/U_{H} varied inversely with the aspect ratio (U_{H} is the mean velocity at the building height).

8:26AM A1.00003 Coupled Convective and Radiative Heat Transfer Simulation for Urban Environments , STEFAN GRACIK, Graduate Student, MOSTAPHA SADEGHIPOUR, Researcher, GEORGE PITCHUROV, Visiting Scholar, JIYING LIU, MOHAMMAD HEIDARINEJAD, Graduate Student, JELENA SREBRIC, Professor, BUILDING SCIENCE GROUP, PENN STATE TEAM — A building's surroundings affect its energy use. An analysis of building energy use needs to include the effects of its urban environment, as over half of the world's population now lives in cities. To correctly model the energy flow around buildings, an energy simulation needs to account for both convective and radiative heat transfer. This study develops a new model by coupling OpenFOAM and Radiance, open source packages for simulating computational fluid dynamics (CFD) and solar radiation, respectively. The model currently provides themo-fluid parameters including convective heat transfer coefficients, pressure coefficients, and solar heat fluxes that will be used as inputs for building energy simulations in a follow up study. The model uses Penn State campus buildings immersed in the atmospheric boundary layer flow as a case study to determine the thermo-fluid parameters around buildings. The results of this case study show that shadows can reduce the solar heat flux of a building's surface by eighty percent during a sunny afternoon. Convective heat transfer coefficients can vary by around fifty percent during a windy day.

8:39AM A1.00004 Modeling and measuring neighborhood scale flow, turbulence, and temperature within Chicago heat island[1] , PATRICK CONRY, ASHISH SHARMA, LAURA LEO, H.J.S. FERNANDO, University of Notre Dame, MARK POTOSNAK, DePaul University, JESSICA HELLMANN, University of Notre Dame — The modeling of urban heat island (UHI) requires a multi-scale approach as it involves numerous physical phenomena spanning a range of scales. We have performed a comprehensive study of Chicago's UHI via coupling of mesoscale Weather Research and Forecasting (WRF) and micro-scale ENVI-met models. The application of the latter model to a Lincoln Park neighborhood and a parallel observational campaign will be the primary focus of this presentation. ENVI-met employs a computational fluid dynamics model to represent heterogeneity of urban areas, providing fine resolution output of UHI dynamics. In the field campaign, two stations located on rooftops of DePaul University buildings were each equipped with a sonic anemometer and vertical array of thermocouples, allowing investigations of spatial variability of flow, turbulent fluxes, and temperature profiles in an urban roughness sublayer. One of these was located above a rooftop garden and the other above a conventional rooftop. Downscaled output from the WRF model or a set of observational data served as initial and boundary conditions for the ENVI-met model. The model's predicative capabilities were assessed through comparison with another set of observational data, and dynamical causes for the model's poor behavior were identified.

[1]Funded by NSF Grant No. 0934592 and ND-ECI

8:52AM A1.00005 Quantifying the effect of inflow variability in RANS simulations of the JU2003 field experiment , CATHERINE GORLE, CLARA GARCIA SANCHEZ, Von Karman Institute for Fluid Dynamics, DAVID PHILIPS, Cascade Technologies, Inc., GIANLUCA IACCARINO, Stanford University — Predicting flow and dispersion in realistic urban canopies is challenging because of the high variability in the governing flow parameters, such as atmospheric conditions and street-level geometrical characteristics. As a result, one deterministic prediction for a specific condition is unlikely to provide an adequate representation of the problem and uncertainty quantification is required to determine confidence bounds on the predictions. Assessing the predictive capability of the resulting model requires validation with field measurements that represent the full complexity of the problem. In this study we present a comparison of the JU2003 field measurements with computational results from RANS simulations performed within an uncertainty quantification framework. The variability in the inflow conditions observed during the field experiment is represented in the simulations, and regions in the urban canopy that are particularly sensitive to this variability are identified. The standard deviation in the results is compared to that observed during the field measurements. Three uncertain variables were considered: the velocity magnitude and direction and the aerodynamic roughness used in the log law that defines the incoming boundary layer profile. A sparse grid Clenshaw-Curtis Stochastic Collocation approach was used, and a polynomial chaos representation of the velocity at different field measurement locations was constructed to extract the mean and standard deviations.

9:05AM A1.00006 ABSTRACT WITHDRAWN –

9:18AM A1.00007 Approximate Deconvolution Large Eddy Simulation of Atmospheric Turbulence in Spectral Space , LEILA NASR AZADANI, ANNE STAPLES, Virginia Tech — Numerical simulations of geophysical turbulence are best performed by large eddy simulation (LES) in which large geophysical scales are solved numerically and effect of physical and dynamical processes accruing at small scales are modeled. Most LES closure models are based on the forward energy cascade from the large scales to the small scales in three-dimensional turbulence. Since, under most conditions, geophysical flows are considered to be approximately two-dimensional turbulent flows, it is questionable to apply these closure models for LES of geophysical flows. Here we present the approximate deconvolution (AD) closure model as a purely mathematical approach, not based on any physics modeling, for LES of turbulent flows. The AD model employs repeated filtering of the filtered variables to obtain an approximation of the unfiltered variables. We apply the AD closure model in the LES of the bartoropic vorticity equation on the sphere in spectral space.

9:31AM A1.00008 Direct Statistical Simulation of a Two-Layer Primitive Equation Model[1] , WANMING QI, BRAD MARSTON, Brown University — Low-order statistics of the large-scale circulation of planetary atmospheres may be directly accessed by solving the equations of motion for the equal-time statistics. We implement such Direct Statistical Simulation of a two-layer primitive equation model by systematic expansion in the cumulants. The first cumulant is the zonally averaged vorticity, divergence, and temperature as a function of latitude and level, and the second cumulant contains information about nonlocal teleconnections. At second order (CE2) the expansion retains the eddy – mean-flow interaction but neglects eddy-eddy interactions and is realizable. Eddy-eddy interactions appear at third (CE3) order, but care must be taken to maintain realizability with a non-negative probability distribution function. The cumulant expansion is conservative, order-by-order, in the total angular momentum, total energy, and mean-squared potential temperature. First and second cumulants accumulated by time-integration of the two-layer primitive equations are compared with those obtained at the fixed points found at CE2 and CE3 levels of approximation. CE2 reproduces qualitative features of the zonal mean general circulation such as the mid-latitude jets. CE3 improves quantitative agreement in the teleconnections.

[1]Supported in part by NSF under Grants No. DMR-0605619 and No. CCF-1048701.

9:44AM A1.00009 Radiative instabilities in vertically sheared rotating stratified flows , CHRISTOPHE MILLET, CEA, DAM, France, FRANÇOIS LOTT, LMD, Ecole Normale Superieure, France — One of the preferential location for the breakdown of balanced dynamics are inertial levels. Across these, balanced disturbances become inertia-gravity waves in the linear approximation. In this work, we analyse how an incident potential vorticity anomaly triggers a baroclinic instability in a rotating stratified fluid with a vertical constant shear. The destabilized character of the wave emission is shown to be associated with the presence of an inertial critical layer that couples a balanced edge wave near the ground and gravity waves aloft. One striking feature of the eigenfunctions is that the gravity wave field appears to have a pronounced asymmetry in meridional wavenumber, with larger amplitudes for horizontal wave vectors pointing toward the warm air. The theoretical predictions for the frequency and growth rate of the normal modes are shown to be in good agreement with the WKB approximation for large Richardson numbers; the latter includes an exponentially small term which captures the radiation feedback in the region below the inertial level.

Sunday, November 24, 2013 8:00AM - 9:31AM –

Session A2 Convection and Buoyancy-Driven Flows I: Numerical Simulations 324 - Hans Johnston, University of Massachusetts - Amherst

8:00AM A2.00001 High Rayleigh number simulations in a cylinderical cell with aspect-ratio 1/3 , ERWIN P. VAN DER POEL, University of Twente, ROBERTO VERZICCO, University of Rome, Tor Vergata, DETLEF LOHSE, University of Twente — The results of DNS simulations of Rayleigh-Bénard convection with Ra up to 10^{12} in a cylindrical geometry with aspect-ratio $1/3$ are presented. The simulations were carried out on a PRACE tier 0 grant illustrating the size of the computational task, which required millions of CPU hours. With $\mathrm{Pr} = 0.7$ these simulations match the new experimental setup build in the "U-boat of Göttingen." We show global quantities such as the heat transport as well as local time-averages. The boundary layer profile and the strength of the large scale circulation are studied and movies of horizontal cross-section of the bulk and the boundary layer are shown.

8:13AM A2.00002 Numerical simulations of high Rayleigh, Prandtl and Schmidt number flows using multiple space/time resolutions , ROBERTO VERZICCO, Università di Roma Tor Vergata, Dipartimento di Ingegneria Industriale, RODOLFO OSTILLA MONICO, ERWIN P. VAN DER POEL, DETLEF LOHSE, University of Twente, Physics of Fluids, PHYSICS OF FLUIDS TEAM — The numerical simulation of passive and active scalars in turbulence becomes more challenging as their diffusivity decreases. In fact, for large Prandtl or Schmidt numbers the Batchelor scale η_T is smaller than the Kolmogorov scale η and, being mesh size taylored to the smallest of the two, the momentum equation is integrated in space and time using unnecessary fine resolutions. This strongly penalizes the computation because, while the scalar dynamics is described by a single equation, the momentum evolves according to a vectorial equation and an elliptic equation for the pressure. Contrary to the intuition, it has observed that even in the case of a flow at Pr=0.7 the resolution needed for the scalar is larger than that of the momentum since the absence of pressure in the equation of the former keeps localized steep gradients. Motivated by the above observation here we show a novel numerical procedure that decouples the space and time resolutions of momentum and temperature and allows to use a refined mesh only for the quantities that need it. We show that, provided every quantity is adequately resolved, the conservation properties of the schemes are retained and at least an order of magnitude reduction of the computational effort is achieved.

8:26AM A2.00003 Effects of Velocity and Temperature Boundary Conditions in Turbulent Thermal Convection , HANS JOHNSTON, UMass Amherst, DAVID GOLUSKIN, Columbia University, CHARLES DOERING, University of Michigan, GLENN FLIERL, MIT — We report on results of high resolution direct numerical simulations of two-dimensional Rayleigh-Bénard convection for Rayleigh numbers up to $Ra = 10^{10}$ in order to study the influence of both temperature and velocity boundary conditions on the turbulent heat transport. In the first scenario, while imposing the no-slip velocity boundary condition, we consider the extreme cases of fixed heat flux (where the top and bottom boundaries are poor thermal conductors) and fixed temperature (perfectly conducting boundaries). Both cases display identical heat transport at high Rayleigh numbers fitting a power law $\nu \approx 0.138 \times Ra^{.285}$ with a scaling exponent indistinguishable from $2/7 = 0.2857\ldots$ above $Ra = 10^7$. The findings are compared and contrasted with results of recent three-dimensional simulations and experiments. In the second scenario we consider the setup originally considered by Rayleigh for calculating conditions for the onset of thermal convection, fixed temperature boundary condition with free-slip velocity boundary conditions. Somewhat surprisingly, at high Rayleigh numbers a strong shear flow develops with periodic "bursting" of the thermal boundary layers. We'll discuss this phenomena and its impact on the heat transport.

8:39AM A2.00004 Solutions to inverse plume in a crosswind problem using a predictor – corrector method , JOSEPH VANDERVEER, YOGESH JALURIA, Rutgers University — Investigation for minimalist solutions to the inverse convection problem of a plume in a crosswind has developed a predictor – corrector method. The inverse problem is to predict the strength and location of the plume with respect to a select few downstream sampling points. This is accomplished with the help of two numerical simulations of the domain at differing source strengths, allowing the generation of two inverse interpolation functions. These functions in turn are utilized by the predictor step to acquire the plume strength. Finally, the same interpolation functions with the corrections from the plume strength are used to solve for the plume location. Through optimization of the relative location of the sampling points, the minimum number of samples for accurate predictions is reduced to two for the plume strength and three for the plume location. After the optimization, the predictor-corrector method demonstrates global uniqueness of the inverse solution for all test cases. The solution error is less than 1% for both plume strength and plume location. The basic approach could be extended to other inverse convection transport problems, particularly those encountered in environmental flows.

8:52AM A2.00005 Time-dependent dynamics of fluid temperature driven by a constant temperature vertical wall in an insulated space , RACHAEL BONNEBAIGT, DAMTP, University of Cambridge, C.P. CAULFIELD, BP Institute & DAMTP, University of Cambridge, P.F. LINDEN, DAMTP, University of Cambridge — We consider the time-dependent flow induced by heating at a vertical wall, held at constant temperature, in a sealed insulated box. Conservation of volume flux, momentum flux, and buoyancy flux give equations for the plume that rises up the wall and for return flow in the ambient fluid. We solve these equations numerically with three different assumptions: a) plume fluid spreading at the ceiling mixes "perfectly" throughout the box down to a first front, leading to two-layer stratification; b) plume fluid spreads at the ceiling with 'zero' mixing into the evolving ambient fluid, leading to continuous ambient stratification; c) the heat transfer coefficient at the wall varies with height according to the classical model of F. J. Bayley (1955 *Proc. I. M. E.* **169** 361-370), i.e. that the Nusselt number is proportional to the one third power of an appropriate Rayleigh number. All schemes reach the same final state: the box reaches the wall temperature and the plume shuts down. We compare the three predictions for the time-dependent ambient temperature distribution with analogue laboratory experiments.

9:05AM A2.00006 Buoyancy-driven flow around $A + B \rightarrow C$ reaction fronts propagating in Hele-Shaw cells: Parabolic flights experiments and numerical simulations , LAURENCE RONGY, Non Linear Physical Chemistry Unit, Universite Libre de Bruxelles (ULB), KERSTIN ECKERT, Institute of Fluid Mechanics, Technische Universitat Dresden, ANNE DE WIT, Non Linear Physical Chemistry Unit, Universite Libre de Bruxelles (ULB) — The dynamics of $A + B \rightarrow C$ reaction fronts is studied under modulated gravitational acceleration by means of a combination of parabolic flight experiments and numerical simulations. During modulated gravity the front position undergoes periodic modulation with an accelerated front propagation under hyper-gravity together with a slowing down under low gravity. The underlying reason for this is an amplification and a decay respectively, of the buoyancy-driven double vortex associated with the front propagation under standard gravitational acceleration, as explained by reaction-diffusion-convection simulations of an $A + B \rightarrow C$ front propagating in a thin layer. Deeper insights into the correlation between grey-value changes in the experimental shadowgraph images and characteristic changes in the concentration profiles are obtained by a numerical simulation of the imaging process.

9:18AM A2.00007 Non-Boussinesq exchange flow over topography , MAZIYAR JALAAL, BORIS STOEBER, Department of Mechanical Engineering, The University of British Columbia, Vancouver, BC, Canada, GREGORY A. LAWRENCE, Department of Civil Engineering, The University of British Columbia, Vancouver, BC, Canada — A series of numerical simulations are performed for the "lock exchange" problem in a two-dimensional duct, where the density ratio of the two phases is varied between 10 and 1000. A finite volume method based on an adaptive Cartesian grid is used with grid refinement in regions of high vorticity and/or density gradient. The physics of the problem is analyzed in detail, including wave formation, disturbance growth and the influence of the density ratio on flow features. The results are compared with laboratory experiments, DNS, and theoretical predictions (single and double -layer shallow water equations). The effects of introducing an obstacle are also investigated.

Sunday, November 24, 2013 8:00AM - 9:57AM –

Session A3 Multiphase Flows I 325 - Gustaf Jacobs, San Diego State University

8:00AM A3.00001 Acceleration of non-Newtonian multiphase flow computations via CPU-GPU platforms , ARTURO FERNANDEZ, North Carolina A&T State University — Modeling of multiphase flow involving non-Newtonian fluids presents special challenges due to the wide range of scales associated with these systems. The reproduction of viscoelastic properties can be done either using constitutive equations derived from kinetic theory or by modeling the dynamics of suspension of macromolecules at the mesoscopic or molecular scale. We present results combining front-tracking with Brownian dynamics simulations, which capture non-Newtonian properties at the mesoscopic scale in a more realistic fashion but at the expense of higher computational cost. We discuss the acceleration of the computations using CPU-GPU platforms. The continuum simulations are carried out in CPUs as usual but the Brownian dynamics simulations are parallelized so they can be performed in GPUs. The examples include the settling of a solid particle in an elastic fluid, the so-called standard case, and the deformation of an elastic drop in a simple shear flow. Good agreement between experimental and numerical results is found.

8:13AM A3.00002 Buoyancy effects on rotation and translation of large particles in turbulent flow , MARGARET BYRON, YIHENG TAO, EVAN VARIANO, University of California Berkeley — We use laboratory experiments to investigate the effects of homogeneous, isotropic turbulence on particles of varying buoyancy, size, and shape. The buoyancy is varied between a specific gravity of 1.001 and 1.05. All particles are roughly 1 cm, which in this flow is close to Taylor's turbulent microscale. We vary the shape to compare spherical particles to non-spherical particles while matching the settling velocity, volume, and/or surface area. Particles are fabricated in custom shapes using transparent hydrogels whose refractive index is close to water. We embed tracers within the particles and use PIV to image the interior of the particle simultaneously with the exterior flowfield of homogeneous isotropic turbulence, generated by two active-grid synthetic jet arrays. We find that the settling velocity of these particles, regardless of shape, is reduced relative to the quiescent settling velocity as predicted by the Clift-Gauvin model. We explore the distribution of rotation rates, as characterized by the variance of angular velocity. We find significant anisotropy in the angular velocities of negatively buoyant particles, which vanishes as particles approach neutral buoyancy. We also see differences in angular velocity distribution between particles of varying eccentricity.

8:26AM A3.00003 Experimental Investigation of Two Phase Fluid Flow and Passive Scalar Mixing around a Periodic Array of Spheres , MAHDI RAMEZANI, SHANKAR SUBRAMANIAM, MICHAEL OLSEN, Iowa State University — Solid-liquid two-phase flow occurs in colloidal suspensions in a variety of applications, from catalytic reactions in chemical plants to bio oil production reactors. Detailed experimental study of these flows can both improve the understanding of the underlying phenomena and also assist in the development of accurate computational modeling. In the presented work, particle image velocimetry (PIV) and planar laser induced fluorescence (PLIF) are used to collect quantitative velocity and scalar field data in a liquid flow containing solid spheres. The velocity and scalar data are collected with sufficient spatial resolution to accurately capture turbulent flow statistics, allowing for precise validation of numerical models. This study is focused on presenting experimental data for periodic arrays of spheres that can be efficiently modeled numerically. For example, one flow geometry investigated consists of a square duct incorporating 5 spheres in line in the axial direction in the middle of the channel. Mixing of a passive scalar as well as the velocity field for this configuration of particles will be presented in the range of Reynolds numbers 50-400 and volume fractions of 5 to 20 percent.

8:39AM A3.00004 A unified model from dense and dilute compressible multiphase flows - application to explosive dispersal[1] , Y. LING, Institute Jean le Rond d'Alembert, Université Pierre et Marie Curie, S. BALACHANDAR, Department of Mechanical and Aerospace Engineering, University of Florida, T.P. MCGRATH, J. ST. CLAIR, Naval Surface Warfare Center, Indian Head Division — Compressible multiphase flows are commonly seen in nature and industrial applications, such as volcanic eruptions and multiphase explosions. A fundamental challenge in modeling of compressible multiphase flows arise from rapid evolution of the volume fraction of the dispersed phase. For example, in multiphase explosions, the volume fraction of particles can change from the close-packing limit to lower than 1% in milliseconds. Since the dominant physics are substantially different in the dense and dilute multiphase flow regimes, the models in the literature for these two regimes are typically different. To accurately simulate compressible multiphase flows involving fast transition from dense to dilute regimes, a novel model that covers both dense and dilute regimes is proposed in this study. A particle volume fraction equation is introduced, which reduces to the particle compaction equation in the dense regime. The present model also includes the added-mass force in the interphase coupling, which has been shown to be important in capturing shock-particle interaction in previous studies. A characteristic analysis of the present model is performed and the effect of added-mass force on acoustic speed of the multiphase system is also analyzed.

[1]AFOSR (FA9550-10-1-0309)

8:52AM A3.00005 A new drag force model based on drift flux for gas-particle two-phase flow , ZHI SHANG, JING LOU, HONGYING LI, IHPC, Singapore — A drag force model was developed to simulate gas-particle two-phase flows. The drag force model was based on the gradients of the drift flux by considering the centrifugal force on the solid particles. According to the gradients of the drift flux, the terminal velocities of the dispersed phase (solid particles) were able to be calculated by the revised gravity. Through the numerical simulations comparing with the experiments and the simulations of the traditional k-ε-Ap and k-ε-kp models, this model was validated.

9:05AM A3.00006 Analysis of turbulent cavitating flow in a micro channel , CHRISTIAN EGERER, STEFAN HICKEL, STEFFEN SCHMIDT, NIKOLAUS ADAMS, Institute of Aerodynamics and Fluid Mechanics, Technische Universität München — Associated with the collapse of vapor cavities is the formation of shock waves and liquid micro-jets, which can lead to the damage of material (cavitation erosion) or even failure of engineering devices, e.g. fuel injectors. We performed Large-Eddy Simulations of the turbulent cavitating flow through a micro channel, resembling a throttle valve commonly found in fuel injectors, at two different operating points with the aim of indentifying such erosion sensitive areas. The underlying numerical method of our flow solver INCA solves the compressible Navier-Stokes equations on a Cartesian adaptive grid for a homogeneous mixture of liquid and vapor in order to account for all relevent physical effects, i.e., compressibility of the liquid-vapor mixture as well as transitional flow and turbulence. The effect of non-represented scales on the represented ones is accounted for by the Adaptive Local Deconvolution Method, a non-linear finite volume scheme for the convective fluxes. We will present a comparison of numerical results with experiments as well as a detailed analysis of the interplay between vortical and cavitation structures. Furthermore, tools enabling the automatic detection of erosion sensitive areas will be discussed and applied.

9:18AM A3.00007 Pressure-driven displacement of a viscoplastic material by a Newtonian fluid[1] , PINAKINARAYAN SWAIN, PHD Student, GEORGE KARAPETSAS, Department of Mechanical Engineering, University of Thessaly, Greece, OMAR MATAR, Department of Chemical Engineering, Imperial College London, South Kensington Campus, SW7 2AZ, KIRTI SAHU, Department of Chemical Engineering, Indian Institute of Technology Hyderabad, Yeddumailaram 502 205, Andhra Pradesh, India — The pressure-driven displacement of a non-Newtonian fluid by a Newtonian fluid in a two-dimensional channel is investigated via a multiphase lattice Boltzmann method using a non-ideal gas equation of state well-suited for two incompressible fluids. We validate the code by comparing the results obtained using different regularized models, proposed in the literature, to model the viscoplasticity of the displaced material. Then, the effects of the Bingham number, which characterises the behaviour of the yield-stress of the fluid and the flow index, which reflects the shear-thinning/thickening tendency of the fluid, are studied. We find that increasing the Bingham number and increasing the flow index increases the size of the unyielded region of the fluid in the downstream portion of the channel and increases the thickness of the residual layer. This in turn decreases the interfacial instabilities and the speed of the propagating finger.

[1]Department of Science and Technology, India

9:31AM A3.00008 Flow of a particulate mixture in a micro-channel , WEITAO WU, Carnegie Mellon University, NADINE AUBRY, Northeastern University, MEHRDAD MASSOUDI, National Energy Technology Laboratory (NETL) — We consider the flow of a mixture of granular materials and a viscous fluid in a micro-channel. We use Mixture Theory to treat this problem as a two-component system. One component (the granular materials) is modeled as a generalized Reiner-Rivlin type fluid, which not only considers the effects of volume fraction but also has a viscosity which depends on the shear rate. The other component (the host fluid) is assumed to behave as a linear viscous fluid. Lift and drag forces exerted by the fluid onto the discrete (solid) particles are taken into account. In order to gain insight into the nature and influence of the various terms in the two phase model we perform a parametric study. Results for the volume fraction and the velocity profiles will be presented.

9:44AM A3.00009 Two-Phase Flow Frictional Characteristics in Porous Wall Bounded Microchannels , EON SOO LEE[1], New Jersey Institute of Technology, JULIE STEINBRENNER[2], University of Colorado, Boulder, CARLOS HIDROVO[3], University of Texas, Austin, KENNETH GOODSON[4], JOHN EATON[5], Stanford University — This presents experimental results from small rectangular channels for fuel cells in which three of the channel walls are smooth, impermeable solid and the fourth wall is a porous gas-diffusion layer. Experiments were performed on a straight 200 by 500 micron by 150 mm long rectangular channel. Three walls of the channel were machined into a solid piece of acrylic. One of the 500 micron wide walls was a commercial Toray carbon paper Gas-Diffusion Layer (GDL) material held in place by a flat sheet of acrylic. Water was forced through the GDL layer from four evenly spaced holes in the flat acrylic piece. A one-dimensional, two-phase flow model was developed which included the effect of air and water flows in both the channel and GDL. The analysis from experimental measurements showed that the product of the friction factor and the gas flow Reynolds number was very nearly a constant, indicating that the model captures the critical physical features of the flow and is useful for the prediction of gas flow rate or pressure drop in a fuel cell microchannel.

[1]Assistant Professor at New Jersey Institute of Technology
[2]Instructor at University of Colorado, Boulder
[3]Assistant Professor at University of Texas, Austin
[4]Professor and Chair of Mechanical Engineering at Stanford University
[5]The Charles Lee Powell Professor at Stanford University

Sunday, November 24, 2013 8:00AM - 9:44AM –

Session A4 Boundary Layers I: Shock Wave Boundary Layer Interaction 326 - Farrukh Alvi, Florida State University

8:00AM A4.00001 The Anatomy of a Shock-Boundary Layer Interaction in Hypervelocity Flow[1]

, ANDREW KNISELY, University of Illinois - Urbana, ANDREW SWANTEK, Argonne National Laboratory, JOANNA AUSTIN, University of Illinois - Urbana — We examine laminar shock-boundary layer interaction over a double wedge geometry in hypervelocity flow. The macroscopic features of this configuration have been shown to be sensitive to the thermochemical energy exchange occurring on a molecular scale. In the current work, an expansion tube is used to accelerate air and nitrogen gas to hypervelocity flow conditions (3.8 km/s, 8.0 MJ/kg) over a 30-55 degree double wedge model. To examine the response of the gas dynamic flow features to real gas effects, we "tune" the chemical composition (O2 content) of the freestream by varying the relative ratio of nitrogen gas and air in the initial test gas. High speed schlieren and chemiluminescence (100k fps) are used to produce overlaid images that visualize the flow structures and identify regions of increased thermochemical activity. These qualitative data are combined with quantitative, pointwise NO vibrational temperature measurements made in the A-X transition band (220–255 nm) to investigate regions of interest such as behind the bow shock and in the shear layer. A transition in bow shock standoff distance and post-shock temperature profiles is identified at 50% O2 content.

[1]The authors would like to acknowledge funding from the U.S. Air Force Office of Scientific Research, grant number FA 9550-11-1-0129, with Dr John Schmisseur as program manager.

8:13AM A4.00002 Response of Hypervelocity Boundary Layers to Global and Local Distortion

, WILLIAM FLAHERTY, JOANNA AUSTIN, University of Illinois — Concave surface curvature can impose significant distortion to compressible boundary layer flows due to multiple, potentially coupled, effects including an adverse pressure gradient, bulk flow compression, and possible centrifugal instabilities. Approximate methods provide insight into dominant mechanisms, however few strategies are capable of treating heat transfer effects and predictions diverge significantly from the available experimental data at larger pressure gradient. In this work, we examine the response of boundary layers to global and local distortions in hypervelocity flows where thermochemical energy exchange has significant impact on boundary layer structure and stability. Experiments are carried out in a novel expansion tube facility built at Illinois. We demonstrate that reasonable estimates of the laminar heat flux augmentation may be obtained as a function of the local turning angle, even at the conditions of greatest distortion. As a model problem to study the evolution of large-scale structures under strained conditions, streamwise vortices are imposed into the boundary layer. The impact of the additional local distortion is investigated. The heat transfer scaling is found to be robust even in the presence of the imposed structures.

8:26AM A4.00003 *A priori* estimates of subgrid-scale terms for LES of shock-boundary layer interactions[1]

, AVINASH JAMMALAMADAKA, FARHAD JABERI, Michigan State University, MICHIGAN STATE UNIVERSITY TEAM — The shock wave interaction with a turbulent boundary layer in high speed flows is very complex and requires high-fidelity numerical methods like direct numerical simulation (DNS) and large-eddy simulation (LES) to capture the flow physics. With the obvious limitations of DNS, we look upon LES as a viable alternative to provide us with an accurate description of shock-boundary layer interaction (SBLI) in high Reynolds number flows. Although there have been some promising results for SBLI, and compressible flows, in general, using LES, there still exists a potential to further improve the accuracy of the numerical model. In this study, *a priori* estimates of various subgrid-scale (SGS) terms in the compressible filtered Navier-Stokes equations are made using highly accurate DNS data for SBLI. The SGS stresses and their components, namely, Leonard, Cross and Reynolds, are examined in various regions of the flow for different shock strengths and filter widths. The backscatter in various regions of the flow was computed and was found to be significant only instantaneously. A term-by-term analysis of the SGS terms in the filtered total energy equations indicated that while each term was significant by itself, the net contribution by all the terms was relatively small and this was indicated in the *a posteriori* analysis as well.

[1]This work is based on the research sponsored by the Air Force Research Laboratory under the agreement number FA8650-06-2-3625.

8:39AM A4.00004 Multi-fidelity numerical simulations of shock/turbulent-boundary layer interaction with uncertainty quantification[1]

, IVAN BERMEJO-MORENO, LAURA CAMPO, Stanford University, JOHAN LARSSON, University of Maryland, College Park, MIKE EMORY, JULIEN BODART, FRANCISCO PALACIOS, GIANLUCA IACCARINO, JOHN EATON, Stanford University — We study the interaction between an oblique shock wave and the turbulent boundary layers inside a nearly-square duct by combining wall-modeled LES, 2D and 3D RANS simulations, targeting the experiment of Campo, Helmer & Eaton, 2012 (nominal conditions: $M = 2.05$, $Re_\theta = 6,500$). A primary objective is to quantify the effect of aleatory and epistemic uncertainties on the STBLI. Aleatory uncertainties considered include the inflow conditions (Mach number of the incoming air stream and thickness of the boundary layers) and perturbations of the duct geometry upstream of the interaction. The epistemic uncertainty under consideration focuses on the RANS turbulence model form by injecting perturbations in the Reynolds stress anisotropy in regions of the flow where the model assumptions (in particular, the Boussinesq eddy-viscosity hypothesis) may be invalid. These perturbations are then propagated through the flow solver into the solution. The uncertainty quantification (UQ) analysis is done through 2D and 3D RANS simulations, assessing the importance of the three-dimensional effects imposed by the nearly-square duct geometry. Wall-modeled LES are used to verify elements of the UQ methodology and to explore the flow features and physics of the STBLI for multiple shock strengths.

[1]Financial support from the United States Department of Energy under the PSAAP program is gratefully acknowledged.

8:52AM A4.00005 Conditional Analysis of a Shock Wave and Turbulent Boundary Layer Interaction[1]

, JUSTINE LI, STEPHAN PRIEBE, M. PINO MARTIN, University of Maryland, College Park — The characterization of the aperiodic cycle of unsteadiness in shock wave and turbulent boundary layer interaction (STBLI) is presented for a 24° compression ramp configuration with a fully turbulent incoming boundary layer at Mach 2.9. In previous work on the direct numerical simulation (DNS) at similar conditions, Priebe and Martn (JFM 2012) found that the aperiodic low-frequency unsteady shock motion is related to the phase of separation bubble growth and collapse. At a reduced computational cost as compared to the DNS, the large eddy simulation (LES) enables the generation of data covering a sufficiently long sampling time to converge the conditional statistics on the phases of the cycle.

[1]This research is supported by AFOSR Grant Number AF/9550-10-1-0164 and by the Department of Defense through the NDSEG Fellowship Program.

9:05AM A4.00006 Characterization of the Shear Layer in a Mach 3 Shock/Turbulent Boundary Layer Interaction

, CLARA HELM, STEPHAN PRIEBE, JUSTINE LI, University of Maryland, PIERRE DUPONT, Aix-Marseille Universite, PINO MARTIN, University of Maryland — The unsteady motion of fully separated shock and turbulent boundary layers interactions (STBLIs) is characterized by an energized low-frequency motion that is two orders of magnitude lower than that of the incoming turbulence. In addition, the spectra shows significant energy content at frequency that is between the characteristic low frequency and the higher frequency motions of the incoming turbulence. The intermediate frequency content is hypothesized to be associated with the existence of Kelvin-Helmholtz type structures, which form in the shear layer downstream of the separation shock and are shed near the reattachment point downstream of the interaction. The current research is concerned with investigating the origins of the intermediate frequencies, and how they may be related to or possibly influence the low-frequency unsteadiness. Specifically, LES data of a Mach 3 STBLI over a 24o ramp are used to estimate convection velocities within the shear layer downstream of the shock. In addition, Brown and Thomas type correlations are used to estimate time and length scales of the eddies in the shear layer. This work is supported by the Air Force Office of Scientific Research under grant AF/9550-10-1-0164.

9:18AM A4.00007 Transient unsteadiness of SWBLI in an axisymmetric geometry , WOUTIJN J. BAARS[1], The University of Melbourne, CHARLES E. TINNEY[2], The University of Texas at Austin — Shock wave boundary layer interactions (SWBLIs) inside an axisymmetric large area ratio nozzle ($M_e = 5.58$) are studied by way of unsteady wall pressure measurements. First, a case of non-transient SWBLI is considered by operating at a nozzle pressure ratio of 28.7, at which a RSS structure forms with trapped annular separation bubbles [Baars et al. AIAA J. 50:1, 2012]. Conditional selection of the data [Erengil and Dolling, AIAA J. 29:5, 1991] resemble similar unsteady features as encountered in nominally 2D interactions. That is, 1) pressures increase in the separated regions as the incipient separation shock translates downstream, and vice versa, which indicates a breathing behavior, and 2) the PDF of the time between shock crossings in the intermittent region is highly skewed, e.g. the shock zero frequency is 33% of the most probable frequency. Secondly, ramping the pressure ratio sweeps the shock system over the transducers and allows the study of transient SWBLI. Time-frequency analyses reveal global features of the unsteady wall signatures, such as low-frequency oscillations in separated regions, and it is identified that nozzle shut-downs are more energetic than start-ups.

[1]Post Doctoral Research Fellow
[2]Assistant Professor

9:31AM A4.00008 Constrained Large-eddy Simulation of Supersonic Turbulent Boundary Layer over a Compression Ramp , LIANG CHEN, ZUOLI XIAO, YIPENG SHI, SHIYI CHEN, Peking University — The mean and statistical quantities as well as the flow patterns of a supersonic turbulent boundary layer over a compression ramp are numerically investigated using the constrained large-eddy simulation (CLES) method. The compression ramp is characterized by a deflection angle of 24. The free-stream Mach number is Ma=2.9, and the Reynolds number based on the inlet boundary layer thickness is Reθ =2300, in accordance with the reference experiment. A rescaling recycling technique is utilized for imposing the inflow boundary. Both the spatial average and the time average methods are employed in the constraint conditions for the Reynolds stresses and heat flux in the near-wall region. The results from CLES are well compared with those from detached-eddy simulation (DES), Reynolds-averaged Navier-Stokes (RANS) simulation, traditional large-eddy simulation (TLES), the experimental and DNS data. It is found that the wall-friction distribution, the wall-pressure distribution, the size of separation bubble, etc., predicted by CLES are in good agreement with the experimental and/or DNS data. Meanwhile, CLES proves to be able to predict the locations of separation and reattachment more accurately than DES, RANS and TLES.

Sunday, November 24, 2013 8:00AM - 9:57AM –

Session A5 CFD I: Immersed Boundary Methods 327 - Shen Xu, Southern Methodist University

8:00AM A5.00001 A sharp, robust, and conservative geometric immersed boundary technique for moving boundaries , PETER BRADY, OLIVIER DESJARDINS, PERRINE PEPIOT, Cornell University — Simulation of solid-fluid systems with complex moving boundaries can be greatly simplified using immersed boundary (IB) methods. IB methods allow for the representation of complex geometries on simple (i.e., Cartesian) meshes, providing an alternative to using a full body-fitted mesh, which often requires an unstructured CFD code and a costly grid-regeneration procedure at every time step. However, using a non body-fitted mesh with IB creates new challenges, including insufficient accuracy in the application of boundary conditions and the potential lack of conservation properties. These challenges are further exacerbated when considering a moving IB. Using a cut-cell IB approach, where the cells that intersect with the solid body are cut such that they become body-fitted, allows for a sharp, discretely conservative IB treatment. With moving geometries, sharp IB methods tend to lack robustness due to the "fresh-cell/dead-cell" problem - the addition or removal of fluid control volumes from the mesh. Rather than relying on an interpolation/smoothing operation to address this issue, a novel semi-Lagrangian geometric transport scheme is used. This fully conservative treatment is verified using the method of manufactured solutions and validated with several complex flows.

8:13AM A5.00002 Derivation of jump conditions for the immersed interface method with a triangular mesh of an interface , GLEN PEARSON, SHENG XU, Southern Methodist University — The immersed interface method is an accurate and efficient Cartesian grid method for solving interface problems. The key idea of the method is to incorporate necessary interface-induced jump conditions into numerical schemes. In this talk, we present an approach to derive the necessary Cartesian jump conditions for the immersed interface method to solve Poisson equations subject to sharp interfaces in 3D. The approach is based on triangular mesh representation of an interface and can easily handle a non-smooth complex interface. We test this approach on Poisson problems with sharp interfaces shaped as spheres, cubes, cylinders and cones. Our results demonstrate second-order accuracy in the infinity norm.

8:26AM A5.00003 The immersed interface method for fluid-solid interaction with boundary condition capturing on triangular meshes[1] , SHENG XU, Math Department, Southern Methodist University — In the immersed interface method, the effect of a rigid solid moving in a fluid is represented as jump conditions incorporated into the discretization of the flow governing equations on a fixed Cartesian grid. In this talk, I present a strategy to numerically compute the required jump conditions toward boundary condition capturing on triangular meshes for solid surfaces. I focus on how to invert a surface gradient operator using a triangular surface mesh to obtain the jump condition of the pressure with desired accuracy. With the boundary condition capturing on triangular surface meshes, the immersed interface method can treat non-smooth solid surfaces. Last, I provide numerical tests to demonstrate the accuracy, efficiency and robustness of the method.

[1]This work is supported by the NSF grant DMS-1320317.

8:39AM A5.00004 Model order reduction of embedded boundary models , MACIEJ BALAJEWICZ, CHARBEL FARHAT, Stanford University — Embedded boundary methods for Computational Fluid Dynamics (CFD) and fluid-structure interaction problems are gaining popularity because they alleviate computational challenges associated with meshing and large wall boundary motions, deformations, and even topological changes. Developing model order reduction methods for computational frameworks based on the embedded boundary method seems however to be challenging. Indeed, most popular model reduction techniques are projection-based and rely on the computation of fluid basis functions based on simulation snapshots. In a traditional body-fitted computational framework, this computation is straightforward because the fluid always occupies the same computational domain. In the embedded computational framework however, deriving global fluid basis functions is problematic a priori because the Eulerian fluid mesh is traversed by the Lagrangian structural mesh. Hence, snapshots collected at different time-instances lose in this case a sense of coherency or consistency. Nevertheless, we demonstrate that this loss of coherency is not a show-stopper for projection-based model reduction based on snapshots.

8:52AM A5.00005 Direct Numerical Simulations of Solid-Fluid Flows Using a Variant of immersed boundary method in Gerris , PEI SHUI, PRASHANT VALLURI, University of Edinburgh, UK, STÉPHANE POPINET, National Institute of Water and Atmospheric research, New Zealand — A novel 3D Immersed Boundary Method simulating fully coupled fluid-solid interaction with 6 degrees-of-freedom (6DOF) movement has been developed under the aegis of the GERRIS code. Any number of fully immersed solids with complex shapes can be considered. A repulsive force which is the sum of all short-range interactions accounts for collisions and ensures that the solids and the wall do not intersect. The solid-fluid solver has been validated against a series of benchmark cases at a wide range of solid Reynolds numbers (0.1 1000) including that of Jeffrey's orbits. In addition, strong hydrodynamic interaction is seen between multiple solids placed in shear flow. The interaction force is being calibrated as a function of relative distance between the solids, drag and lift forces and will be presented in the conference. In the inertial regime, the problem of migrating dense spheres under Poiseuille flow has been quantitatively validated against simulation results and experiments of Yu et al (2004). There are however some minor divergences in results for migration of neutrally buoyant spheres, but are in full quantitative agreement with experiments of Jeffrey (1989).

9:05AM A5.00006 An efficient boundary condition enforced-immersed boundary method for thermal flows with heat flux condition , WEIWEI REN, CHANG SHU, WENMING YANG, National University of Singapore, YU CHEN, Lloyd's Register Global Technology Center — An efficient boundary condition enforced-immersed boundary method (IBM) is proposed in this work for thermal flows involving complex geometries. By treating the heated immersed boundary as a series of heat sources/sinks, Peskin's original IBM has been extended to heat transfer problems with Neumann condition (heat flux) for the temperature field. The main feature of the present approach is to accurately satisfy the energy equation and its boundary conditions through a heat flux correction procedure, which is performed by introducing a heat source/sink term into the energy equation. The heat source/sink is evaluated from the offset boundary heat flux, which is generated from the difference between the normal temperature derivative in the given Neumann condition and the calculated derivative when the boundary effect is not considered. The present solver has proven to be of second order accuracy through a numerical analysis. Its capability and efficiency have also been validated by applying it to numerical examples like forced convection over a stationary heated circular cylinder and natural convection in a horizontal concentric and eccentric annulus between two circular cylinders, from which good agreements with the established data have been achieved.

9:18AM A5.00007 An Immersed Boundary-Lattice Boltzmann Method for Simulating Particulate Flows , BAILI ZHANG, MING CHENG, JING LOU, Institute of High Performance Computing — A two-dimensional momentum exchange-based immersed boundary-lattice Boltzmann method developed by X.D. Niu et al (2006) has been extended in three-dimensions for solving fluid-particles interaction problems. This method combines the most desirable features of the lattice Boltzmann method and the immersed boundary method by using a regular Eulerian mesh for the flow domain and a Lagrangian mesh for the moving particles in the flow field. The non-slip boundary conditions for the fluid and the particles are enforced by adding a force density term into the lattice Boltzmann equation, and the forcing term is simply calculated by the momentum exchange of the boundary particle density distribution functions, which are interpolated by the Lagrangian polynomials from the underlying Eulerian mesh. This method preserves the advantages of lattice Boltzmann method in tracking a group of particles and, at the same time, provides an alternative approach to treat solid-fluid boundary conditions. Numerical validations show that the present method is very accurate and efficient. The present method will be further developed to simulate more complex problems with particle deformation, particle-bubble and particle-droplet interactions.

9:31AM A5.00008 Algorithmic improvements for accurate force prediction in diffusive-interface direct-forcing immersed boundary method[1] , XING ZHANG, XIAOJUE ZHU, GUOWEI HE, LNM, Institute of Mechanics, Chinese Academy of Sciences — We detail some algorithmic modifications to the diffusive-interface direct-forcing immersed boundary method of Ulhmann (JCP 209(2005) 448-476). The prediction of local hydrodynamic force can be improved by the following two measures taken. First, by using a "force correction" strategy, the fluid penetration near the immersed boundary is significantly reduced. However, the "force correction" method also leads to large spurious oscillation in the Lagrangian force. This is problematic since the Lagrangian force also represents the local hydrodynamic surface force in some fluid-structure-interaction (FSI) simulations. By perform an additional filtering (smoothing) step to the Lagrangian force, it is found that the oscillation can be largely suppressed. Accurate prediction of local surface force by the proposed method is demonstrated using the case of 2D flow over a circular cylinder.

[1]This work was supported by National Natural Science Foundation of China (Grant Nos. 11232011, 11021262 and 11023001).

9:44AM A5.00009 An implicit immersed boundary method for moving body problems in curvilinear coordinates , LAURA NICOLAOU, SEO YOON JUNG, TAMER ZAKI, Imperial College London — A robust immersed boundary method for flow in complex geometries is presented. No-slip conditions are enforced via momentum forcing and mass conservation at the immersed boundary is satisfied via a mass source term developed for moving bodies. Stability is shown to depend on the temporal discretization of the momentum forcing, as inconsistencies between the forcing and the intermediate velocity equations introduce errors near the boundary. An iterative method to compute the forcing term implicitly is proposed, which reduces the errors at the boundary and enhances stability. The convergence of the iterative method, second-order accuracy and enhanced stability of the scheme are demonstrated in a number of test cases. In addition, the proposed mass source term accurately accounts for the movement of the boundary, and reduces the spurious force oscillations which arise in IB simulations of moving body problems. The method is developed for use in a generalised curvilinear system, which lends itself to a wide range of complex flow problems.

Sunday, November 24, 2013 8:00AM - 9:57AM –

Session A6 Microfluids: Mixing 328 - Igor Mezic, University of California, Santa Barbara

8:00AM A6.00001 Synergetic Fluid Mixing from Viscous Fingering and Alternating Injection , BIRENDRA JHA, LUIS CUETO-FELGUEROSO, RUBEN JUANES, Massachusetts Institute of Technology — We study mixing of two fluids of different viscosity in a microfluidic channel or porous medium. We show that the synergetic action of alternating injection and viscous fingering leads to a dramatic increase in mixing efficiency at high Peclet numbers. Based on observations from high-resolution simulations, we develop a theoretical model of mixing efficiency that combines a hyperbolic mixing model of the channelized region ahead, and a mixing-dissipation model of the pseudo-steady region behind. Our macroscopic model quantitatively reproduces the evolution of the average degree of mixing along the flow direction, and can be used as a design tool to optimize mixing from viscous fingering in a microfluidic channel.

8:13AM A6.00002 The Impact of Miscible Viscous Fingering on Mixing , JANE CHUI, PIETRO DE ANNA, RUBEN JUANES, Massachusetts Institute of Technology — Viscous fingering is a hydrodynamic instability that occurs when a less viscous fluid displaces a more viscous one. Instead of progressing as a uniform front, the less viscous fluid forms fingers that vary in size and shape to create complex patterns. The interface created from these patterns affects mixing between the two fluids, and therefore is of critical importance in applications such as enhanced oil recovery and microfluidics. This work focuses on how the evolution of the fingering interface affects mixing between two miscible fluids, specifically in a radial configuration. We measure the local concentration field temporally and spatially with the use of a fluorescent tracer in the injected fluid, and with this high resolution information are able to calculate various measures of mixing, such as mixing efficiency, scalar dissipation rate, and the areal mixing zone for different fluid injection rates and various viscosity ratios. We propose a scaling theory based on experimental observations for the growth of the mixing zone and the overall rate of mixing.

8:26AM A6.00003 Diffusive and inertial instabilities during miscible fluid thread formation in microgeometries[1] , THOMAS CUBAUD, SARA NOTARO, Stony Brook University — We study the formation and stability of miscible fluid threads having large difference in viscosity using hydrodynamic focusing sections. Miscible core annular flows are useful for transporting viscous materials and can be destabilized for enhancing mass transfer. Here, we delineate phase-diagrams of the formation of miscible threads from low to large viscosity contrasts with various diffusion coefficients. Depending on fluid properties and flow rates of injection, microflows are classified into diffusive, stable, and inertial regimes. For low Péclet numbers, we examine threads dynamics when diffusive effects strongly influence flow structures. Another regime is investigated for moderate Reynolds numbers where small threads are rapidly destabilized in the inertial flow field of the sheath fluid at the junction.

[1]This work is supported by NSF (CBET- 1150389)

8:39AM A6.00004 Diffusion Effects on the Chaotic Fluid Mixing for AC Electrothermal Flows by Blinking Vortices , SOPHIE LOIRE, IGOR MEZIC, UCSB — We present a computational study of AC electrothermal chaotic mixing using blinking of asymmetric electrothermal vortices. Electrothermal flows are modeled by finite element method using COMSOL software based on an enhanced electrothermal model. We use the mix-norm on numerical trajectory simulations to evaluate mixing at different scales including the layering of fluid interfaces by the flow, a keypoint for efficient mixing. The blinking vortices method greatly improve mixing efficiency. The effect of blinking frequency and particle size is studied. A large influence of diffusion on the mixing efficiency is observed as well as on the optimal blinking frequency.

8:52AM A6.00005 AC Electrokinetic 3D blinking micro-mixer , MARIN SIGURDSON, SOPHIE LOIRE, MARKO BUDISIC, IGOR MEZIC, UC Santa Barbara — An AC electrokinetic 3D mixer is presented, which has the potential to accelerate bio-reactions in both lab-on-a-chip and microplate formats. AC voltage across electrodes on a micro-chamber floor generate vortices in the buffer via the electrothermal effect, which is particularly effective for ionic buffers used in bioassays. Controlling these vortices over time, for example, by periodic switching between overlapping vortices (blinking), creates time dependent 3D chaotic mixing. This mixing was studied via 2 methods. First, the full 3D, 3 component velocity field was measured with our original Proper Orthogonal Decomposition PIV method for each vortex configuration. These velocity fields were then used to numerically evaluate mixing predictive parameters such as ergotic quotient and fluid layering. These parameters help identify regions of good and poor mixing, aiding electrode shape design. Second, mixing of low and high diffusivity particles was optically evaluated, through the Mix Variance Coefficient. The blinking pattern and frequency was then optimized to yield the fastest mixing for each case. Finally, work is underway to demonstrate reaction rate acceleration on the order of 10 fold as the result of this mixing.

9:05AM A6.00006 Effective mixing strategies with microbubble streaming flows , CHENG WANG[1], BHARGAV RALLABANDI, LIN GUO, SASCHA HILGENFELDT, Department of Mechanical Science and Engineering, University of Illinois at Urbana-Champaign — Homogeneous mixing of chemical/biological samples and reagents is one of the essential preparation steps for lab-on-a-chip systems. As effective Stokes flows driven by fast time scale oscillatory flows, microbubble streaming flows are a tool uniquely positioned between passive and active mixing approaches. Guided by thorough theoretical understanding of the flows and of micromixing itself, we investigate various designs of microbubble mixers, employing two key strategies: (a) introducing controlled unsteadiness in the acoustic driving pattern, e.g. by duty-cycling and driving frequency modulation, and (b) optimizing the arrangement of multiple bubbles, such as the number, position, and orientation of the microbubbles, particularly to generate 3D chaotic flow patterns. Both of these approaches significantly improve mixing over that of previous steady 2D bubble micro-mixers, and the strategies can be combined for greater effect.

[1]Current address: Department of Mechanical and Aerospace Engineering, Missouri University of Science and Technology

9:18AM A6.00007 Microfluidic mixing using orbiting magnetic microbeads[1] , MATTHEW BALLARD, DREW OWEN, WENBIN MAO, PETER HESKETH, ALEXANDER ALEXEEV, Georgia Institute of Technology, Atlanta, Georgia — Using three-dimensional simulations and experiments, we examine mixing in a microfluidic channel that incorporates a hybrid passive-active micromixer. The passive part of the mixer consists of a series of angled parallel ridges lining the top microchannel wall. The active component of the mixer is made up of microbeads rotating around small pillars on the bottom of the microchannel. In our simulations, we use a binary fluid lattice Boltzmann model to simulate the system and characterize the microfluidic mixing in the system. We consider the passive and active micromixers separately and evaluate their combined effect on the mixing of binary fluids. We compare our simulations with the experimental results obtained in a microchannel with magnetically actuated microbeads. Our findings guide the design of an efficient micromixer to be used in sampling in complex fluids.

[1]Financial support from NSF (CBET-1159726) is gratefully acknowledged.

9:31AM A6.00008 ABSTRACT WITHDRAWN –

9:44AM A6.00009 Mixing and transport by ciliary carpets , YANG DING, University of Southern California, JANNA NAWROTH, California Institute of Technolog, MARGARET MCFALL-NGAI, University of Wisconsin, Madison, EVA KANSO, University of Southern California — Cilia are hair-like micro-structures observed on surfaces of many biological systems such as the human lungs. Cilia usually beat asymmetrically in a coordinate manner and serve for flow generation and sensing. Here, we use a 3D computational model to study the fluid transport and mixing due to the beating of an infinite array of cilia. In accord with recent experiments, we observed two distinct regions: a fluid transport region above the cilia and a fluid mixing region below the cilia tip. We examined the effect of the metachronal wave (due to phase differences between neighboring cilia) on the net flow and mixing rate. We found that the metachronal wave can enhance both transport and mixing rate of the fluid, often simultaneously. Our results suggest that the simultaneous enhancement in fluid transport and mixing is due to the enhancement in shear flow. As the flow above the cilia increases, shear rate in the fluid increases and such shear enhances stretching, which is an essential ingredient for mixing. Estimation of the time scale of the mixing indicates that, compared to diffusion, the mixing due to the cilia beat may be significant or even the dominate way of distributing molecules in some biological systems.

Sunday, November 24, 2013 8:00AM - 9:57AM –

Session A7 Microfluids: Interfaces and Wetting I 329 - Kenny Breuer, Brown University

8:00AM A7.00001 New thermal-sensitive superhydrophobic material , RAPHAELE THEVENIN, PMMH Laboratory, ESPCI / Ladhyx, Ecole Polytechnique - France, ZI LIANG WU, PATRICK KELLER, Institut Curie - France, ROBERT E. COHEN, Chemical Engineering, MIT - USA, CHRISTOPHE CLANET, Ladhyx, Ecole Polytechnique/PMMH Laboratory, ESPCI - France, DAVID QUERE, PMMH Laboratory, ESPCI / Ladhyx, Ecole Polytechnique - France — Roughness of superhydrophobic surfaces plays a crucial role in the wetting properties of these surfaces. We propose to modify the roughness of a surface by applying external stimuli to change its wetting properties. In this spirit, we study superhydrophobic surfaces consisting of arrays of micro-pillars made with a liquid crystal elastomer. These liquid crystals change their orientation when heated, so that the height of the pillars decreases when the surface temperature increases; and this is perfectly reversible. We study the impact of such a thermal-actuation on the static and dynamic wetting properties of such surfaces and show superhydrophobicity of this new material can indeed be tuned using temperature stimuli.

8:13AM A7.00002 A generalised view of high frequency substrate vibration induced wetting (Acoustowetting) , OFER MANOR, Chemical Engineering, Technion - Israel Institute of Technology, Haifa, Israel, AMGAD REZK, JAMES FRIEND, LESLIE YEO, Micro/Nanophysics Research Laboratory, RMIT University, Melbourne, Victoria, Australia — High frequency surface vibrations, at frequencies comparable to the HF and VHF radio frequencies O(1–100 MHz), may be used for generating flow at the micron and submicron scale. Such high frequency vibrations are generated by piezoelectric actuators that transfer electric signals to kinetic energy, exciting different types of flow regimes when in contact with viscous fluids that are known in general as acoustic flow. Here we unravel a recently found wetting mechanism, observed in laboratory under the excitation of high frequency vibrations in the form of piston-like substrate motion and surface acoustic waves (SAWs). Wetting is excited by introducing acoustic flow layer of usually submicron thickness near the three phase contact line of liquid/solid systems. This wetting effect further gives rise to various peculiarities including film spreading at different directions according with periodic stability of the film thickness, formation of wave pulse trains, SAW diffraction induced film fingering, etc. We show high frequency vibration induced wetting is governed by a generalised theory that predicts the various physical peculiarities observed.

8:26AM A7.00003 Dynamics of Wetting of Ultra Hydrophobic Surfaces , ALIREZA MOHAMMAD KARIM, University of California, Los Angeles (UCLA), JEONG-HYUN KIM, JONATHAN ROTHSTEIN, University of Massachusetts, Amherst, PIROUZ KAVEHPOUR, University of California, Los Angeles (UCLA), MECHANICAL AND INDUSTRIAL ENGINEERING, UNIVERSITY OF MASSACHUSETTS, AMHERST COLLABORATION — Controlling the surface wettability of hydrophobic and super hydrophobic surfaces has extensive industrial applications ranging from coating, painting and printing technology and waterproof clothing to efficiency increase in power and water plants. This requires enhancing the knowledge about the dynamics of wetting on these hydrophobic surfaces. We have done experimental investigation on the dynamics of wetting on hydrophobic surfaces by looking deeply in to the dependency of the dynamic contact angles both advancing and receding on the velocity of the three-phase boundary (Solid/Liquid/Gas interface) using the Wilhelmy plate method with different ultra-hydrophobic surfaces. Several fluids with different surface tension and viscosity are used to study the effect of physical properties of liquids on the governing laws.

8:39AM A7.00004 Dynamic wetting at the nanoscale[1] , GUSTAV AMBERG, Dept of Mechanics, Linné FLOW center, KTH, Stockholm, Sweden, YOSHINORI NAKAMURA, Dept of Mechanical Eng, the University of Tokyo, Japan, ANDREAS CARLSON, Dept of Mechanics, Linné FLOW center, KTH, Stockholm, Sweden, JUNICHIRO SHIOMI, Dept of Mechanical Eng, the University of Tokyo, Japan — Although the capillary spreading of a drop on a dry substrate is well studied, the physical mechanisms that govern the dynamics remain challenging. Here we study the dynamics of spreading of partially wetting nano-droplets, by combining molecular dynamics and continuum simulations. The latter accounts for all the relevant hydrodynamics, i.e. capillarity, inertia and viscous stresses. By coordinated continuum and molecular dynamics simulations, the macroscopic model parameters are extracted. For a Lennard-Jones fluid spreading on a planar surface, the liquid slip on the substrate is found to be crucial for the motion of the contact line. Evaluation of the different contributions to the energy transfer shows that the liquid slip generates dissipation of the same order as the bulk viscous dissipation or the energy transfer to kinetic energy. We also study the dynamics of spreading on a substrate with a periodic nanostructure. Here it is found that a nanostructure with a length scale commensurate with molecular size completely inhibits the liquid slip. This reduces the spreading speed by about 30%.

[1]This work is partially supported by the Japan Science and Technology Agency, CREST, the Swedish Research Council, the Japan Society for the Promotion of Science, and the Sasakawa foundation.

8:52AM A7.00005 Hard-sphere and inertial effects in colloidal liquid-gas systems , ANDREAS NOLD, BEN GODDARD, SERAFIM KALLIADASIS, Department of Chemical Engineering, Imperial College London, UK — Hard-sphere effects at a liquid-gas contact line on chemically heterogeneous substrates are studied employing density functional theory. For dynamic colloidal liquid-gas systems, a novel extension of dynamic density functional theory (DDFT) to include inertial effects [1,2] is introduced and numerical results for the motion of colloidal droplets are presented. In particular, we present results for the motion of the contact line between a substrate, a colloidal liquid and a colloidal gas phase. The link between the modelling approaches for dynamic colloidal systems using DDFT and the Navier-Stokes equations for molecular systems is discussed.

[1] Goddard, Nold, Savva, Yatsyshin and Kalliadasis, 2013, Unification of dynamic density functional theory for colloidal fluids to include inertia and hydrodynamic interactions: derivation and numerical experiments, J. Phys.: Condens. Matter, 25, 035101;
[2] Goddard, Nold, Savva, Pavliotis, Kalliadasis, 2012, General dynamical density functional theory for classical fluids, PRL, 109, 120603.

9:05AM A7.00006 Electrostatic line tension resulting from fluid-fluid interfacial deformation , AARON DOERR, STEFFEN HARDT, Institute of Nano- and Microfluidics, Tech. Univ. Darmstadt, Darmstadt, Germany — We investigate the deformation of a fluid-fluid interface due to osmotic and electrostatic forces in the three-phase contact region between an electrolyte, a non-polar fluid, and a dielectric solid substrate with both semi-analytical and numerical methods. It is shown that the interfacial deformation decays exponentially as a function of distance from the three-phase contact line, consistent with a well-defined macroscopic contact angle. As a consequence and on a sufficiently large scale of observation, the physical situation may be modeled by extrapolating the macroscopic interfacial shape to the solid substrate, while the energetic contributions associated with the microscopic configuration near the three-phase contact line may be accounted for by an excess energy per unit length of contact line, an electrostatic line tension. A comparison of the semi-analytical model to numerical calculations is used to examine its limits of quantitative applicability. At the same time, it is demonstrated that beyond those limits the model still qualitatively agrees with the numerical results, corroborating its usefulness for understanding the physics close to the three-phase contact line on large as well as on small observation length scales.

9:18AM A7.00007 ABSTRACT WITHDRAWN –

9:31AM A7.00008 Velocity Slip on Curved Surfaces , WEIKANG CHEN, RUI ZHANG, JOEL KOPLIK, City College of New York — The Navier boundary condition for velocity slip on flat surfaces, when expressed in tensor form, is readily extended to surfaces of any shape. We test this assertion using molecular dynamics simulations of flow in linear channels with flat and curved walls and for rotating cylinders and spheres, all for a wide range of solid-liquid interaction strengths. We find that the slip length as conventionally measured at a flat wall in Couette flow is the same as that for all other cases with curved and rotating boundaries, provided the atomic interactions are the same. These results support the idea that the slip length is a material property, transferable between different flow configurations.

9:44AM A7.00009 Measurements of a high-speed receding contact line on a hydrophobic surface , JOONSIK PARK, KENNETH S. BREUER, Brown University — We report on the behavior of a non-equilibrium receding contact line generated by the rapid pinchoff of a liquid bridge extending between a capillary tube and a smooth hydrophobic substrate. The motion of the contact line is measured from below using a high-speed camera (10kfps). Three stages are identified according to the power-law scaling between the droplet radius and the receding time. In the initial non-equilibrium phase, the power-law exponent scales with the retraction speed of the tube. As the contact angle approaches the minimum receding angle, the contact line retreats according to a universal power-law exponent. In the final stage, close to the pinch-off, the receding contact angle increases to the minimum equilibrium contact angle and the power-law exponent decreases. The variation of the behavior is measured as a function of the fluid properties (viscosity, elasticity and surface tension), the tube retraction speed, and the substrate properties (hydrophobicity and surface roughness).

Sunday, November 24, 2013 8:00AM - 9:44AM –

Session A8 Particle-Laden Flows I: Liquid-Solid Flows 330 - Andrea Prosperetti, Johns Hopkins University

8:00AM A8.00001 Natural particles in turbulent aquatic environments: when do they differ from fluid parcels? , EVAN VARIANO, LAURA MAZZARO, MARGARET BYRON, RACHEL ALLEN, IAN TSE, University of California Berkeley — We explore if, and when, natural particles act differently than fluid parcels. We focusing on the case of aquatic environments with high-Reynolds number turbulence. The particles of interest in such environments are suspended sediment (often in the form of porous aggregates), plankton, and droplets (such as oil droplets). We use laboratory experiments (described below) and literature review to build a description of the cases in which these particles display kinematics different than that of fluid parcels which passively follow the flow. Specifically, we explore the effect of size, shape, and density on settling, diffusion, and clustering. The goal for each case is specify critical values at which behavior departs from that of fluid parcels. The laboratory experiments we use employ a simple 3D particle-tracking camera (based on the defocusing principle by Willert and Gharib [1992] and currently used in "V3V" velocimetry). With this, we study a variety of particles in laboratory flow. These data, and the analysis presented herein, is a precursor to studies conducted directly in estuarine environments. Willert, C. E. and Gharib, M. 1992 Three-dimensional particle imaging with a single camera. Experiments in Fluids, 12(6), 353-358.

8:13AM A8.00002 The motion of spherical particles falling in a cellular flow field[1] , ELISABETH GUAZZELLI, GILLES BOUCHET, LAURENCE BERGOUGNOUX, Aix-Marseille Univ., CNRS, IUSTI UMR 7343, GEP TEAM — The objective of the present study is to understand the influence of turbulence on the settling of particles under the action of gravity. This effect is intimately related to the interactions of particles with local spatial structures of the flow, e.g. large vortices. These vortical structures have a significant effect on the local particle transport and concentration. We present a jointed experimental and numerical study to examine these issues. The two-dimensional model experiment uses electroconvection to generate a two-dimensional arrays of controlled vortices which mimic a simplified turbulent flow. Particle image-velocimetry or tracking are used to examine the motion of the particles within this vortical flow. The numerical simulation is inspired by the model developed by Maxey (Phys. Fluids 30, 1915, 1987).

[1]This work was undertaken under the auspices of ANR-12-BS09-0017-01, CNRS-PICS05848, ANR-11-LABX-0092, and ANR-11-IDEX-0001-02.

8:26AM A8.00003 Modeling the rheology of concentrated fluid-sediment mixtures , ALLISON PENKO, JULIAN SIMEONOV, JOSEPH CALANTONI, Naval Research Laboratory — Common formulations to model the rheology in highly concentrated fluid-sediment mixtures include using a velocity damping function or an enhanced effective viscosity dependent on the local sediment concentration. Here, we implemented a three-dimensional mixture theory model (SedMix3D) to compare the results from the two different rheological formulations. The former assumes the mixture behaves as a Newtonian fluid and exponentially damps the mixture velocity dependent on the magnitude of the local sediment concentration. The latter formulation treats the mixture as a visco-plastic in which an enhanced effective viscosity depends on the Reynolds stresses and the critical stress of the sediment. SedMix3D treats a fluid-sediment mixture as a continuum by employing closures for the bulk parameters of the mixture (diffusion, hindered settling velocity, and effective viscosity) to simulate three-dimensional, bottom boundary layer flow over dynamic sediment beds. The two different rheologies were tested with simulations of an avalanching sediment pile forced with gravity only and simulations of oscillatory flow over rippled sand beds. Comparisons of the simulated hydrodynamics and sediment dynamics resulting from the two rheologies will be presented.

8:39AM A8.00004 Numerical modeling of bidensity suspensions in gravity-driven, thin-film flows , JEFFREY WONG, UCLA, SUNGYON LEE, Texas A&M, ALIKI MAVROMOUSTAKI, ANDREA BERTOZZI, UCLA — We present an equilibrium model for bidisperse suspensions consisting of negatively buoyant particles with two different densities that flow down an incline. In the case of monodisperse suspensions, it has been shown experimentally and theoretically that particles either settle to the substrate or accumulate at the fluid surface (these are referred to as the "settled" and "ridged" regimes, respectively), depending on the inclination angle and total particle volume fraction. When there are two species, the heavier particles settle to the substrate while the lighter particles exhibit a similar transition between settled and ridged regimes, which now also depends on the relative concentration. We investigate the model numerically and compare the predicted transition between settled and ridged regimes with experimental results. In addition, we discuss the effect of shear-induced self-diffusion in the model, which leads to some mixing of the particle layers.

8:52AM A8.00005 Erosion and transport of particulates by forced jet impinging jet on a mobile sediment bed[1] , KYLE CORFMAN, RAHUL MULINTI, KENNETH KIGER, University of Maryland — The work reports on the erosion and suspended flux characteristics of a forced impinging jet, as a prototypical surrogate to better understand the problem of rotorcraft brownout. Coherent vortex rings are generated through oscillatory forcing of a vertical impinging jet onto a sediment bed. Early in the flow development, annular ripple dunes are formed and steadily grow, the wavelength and growth rate depending largely on the particle size and flow conditions. In order to provide a reliable prediction of erosion for more realistic flows, such as those found in rotorcrafts, a parametric study was performed for several particle sizes and mixtures. PTV is used to correlate vertical and horizontal fluxes with resulting changes in the ground profiles. A single-phase PIV study detailing the changes in the vortex ring characteristics after the bed has reached a stable erosion pattern is also reported.

[1]This work is supported by the AFSOR under grant FA95500810406.

9:05AM A8.00006 Collision Driven Particle Dynamics Simulations for Analyzing Flows of Particulate Sprays and Jets , DEBANJAN MUKHERJEE, TAREK ZOHDI, University of California, Berkeley — This work presents a detailed overview of the development of a computer simulation tool based on neighbor-list collision driven particle dynamics to investigate the flow of particulate sprays and jets. A detailed discussion of a hierarchical modeling approach to represent coupled, multi-physical phenomena through simple models for underlying physical interactions is presented. The models are based on the concept of individual "particles" or "discrete elements" - which could be actual particles in some applications, and a meso-scale idealized computational unit in others. Particularly, the work focuses on the overall flow behavior in the presence of collisions and interactions with surrounding fluid, and representative simulation examples are presented to illustrate the dispersed particle ensemble dynamics. The simulations were found to be reasonable in performance time using readily available computational resources. From the perspective of engineering software development, the work also briefly addresses the issue of simulation architecture and user front-end. Since this is part of an ongoing research, the status of current development, future research directions, and possibilities of open collaborations both in terms of simulation development and applications will be addressed

9:18AM A8.00007 Pop up height of buoyant spheres , TADD TRUSCOTT, RANDY MUNNS, Brigham Young University, SPLASH LAB TEAM — We examine the rising and surface breaching dynamics of buoyant spheres released at varying depths beneath the free surface in water over a range of Reynolds numbers (Re $= 3 \times 10^4$ to 5×10^5 using high-speed imaging and particle image velocimetry. Buoyant spheres of sufficient speed pop up out of the free surface in varying ways depending on the conditions under the surface. Altering the release depth reveals varying exit angles, velocities, accelerations, and pop up heights at surface exit. Vortex shedding prior to free surface exit causes decelerations contributing to the variation in exit velocities and resulting pop up heights. Through a comprehensive study the phenomenon is extremely predictable. At lower Re, spheres released from shallow depths result in greater accelerations, velocities and pop up heights at the free surface compared to lower pop up heights when released from deeper depths (contrary to intuition). As the depth of release is increased the pop up height oscillates between a maximum and minimum. This is directly related to the proximity of the shed vortex to the free surface. For spheres of greater Re, pop up height increases linearly with release depth, demonstrating continued accelerations at free surface exit.

9:31AM A8.00008 Shock Dynamics for particle-laden thin film , LI WANG, ANDREA BERTOZZI, Department of Math, UCLA — We study the shock dynamics for a recently proposed system of conservation laws (Murisic et. al [J. Fluid Mech. 2013]) describing gravity-driven thin film flow of a suspension of particles down an incline. When the particle concentration is above a critical value, singular shock solutions can occur. We analyze the Hugoniot topology associated with the Riemann problem for this system, describing in detail how the transition from a double shock to a singular shock happen. We also derive the singular shock speed based on a key observation that the particles pilling up at the maximum packing fraction near the contact line. These results are further applied to constant volume case to generate a rarefaction-singular shock solution. The particle/fluid front are shown to move linearly to *the leading order* with time to the one-third power as predicted by the Huppert solution for clear fluid.

Sunday, November 24, 2013 8:00AM - 9:57AM –

Session A9 Instability: Interfacial and Thin-Film I 333 - Gary Settles, Pennsylvania State University

8:00AM A9.00001 Simultaneous measurement of a thickness and wave velocity of a liquid film flow, by using a single-tip optical fiber probe , HAJIME FURUICHI, Graduate School of Engineering, Shizuoka University, TAKAYUKI SAITO, Research Institute of Green Science and Technology, Shizuoka University — We developed a new measurement technique for a liquid film flow, using a single-tip optical fiber probe (S-TOP). The measurement method for a liquid-film thickness is as follows; the S-TOP with a tapered tip was installed parallel to the main stream, and detected a wavy surface; after processing the probe signals, liquid phase fractions were calculated in every installed position of the S-TOP. Moreover, we calibrated the experimental results via our original 3D-ray-tracing numerical-simulation. Analyzing the simulated signals, we found the relationship among the liquid phase fractions, the installed positions and the wave heights. Wave velocities were accurately measured through our original micro-fabricated S-TOP that has two optical-sensors. The experimental and numerical analyses were executed in order to deeply understand the complex signals of the S-TOP. Finally, the simultaneous measurement technique of the thickness and wave velocity was demonstrated. When the liquid phase fraction was 0.52, the installed-position equals the average film thickness. The wave velocity was measured based on the event time of each sensor touching the film surface. A difference in the results of the velocity between the S-TOP and the visualization was less than 10 percent.

8:13AM A9.00002 Nonlinear interfacial dynamics in stratified multilayer channel flows[1] , DEMETRIOS PAPAGEORGIOU, EVANGELOS PAPAEFTHYMIOU, GRIGORIOS PAVLIOTIS, Imperial College London, INTERFACIAL FLUID DYNAMICS COLLABORATION — Viscous immiscible pressure-driven multilayer flows in channels are investigated using a combination of modelling, analysis and computations. Three stratified layers with two internal interfaces are considered and long wave theory is used to derive a coupled system of Benney-type equations containing a small parameter that cannot be scaled out. A consistent system of coupled weakly nonlinear equations is developed and two canonical cases are identified in the absence and presence of inertia, respectively. The system supports instabilities not found in single long-wave equations including, transitional instabilities due to a change of type of the nonlinearities from hyperbolic to elliptic, kinematic resonance instabilities, and long-wave instabilities induced by an interaction between nonlinearity and surface tension. In contrast to two-layer systems instabilities leading to nonlinear traveling waves are possible even at zero Reynolds number. When inertia is present the systems become general coupled Kuramoto-Sivashinsky type equations. Numerical experiments produce dynamics including traveling, time-periodic traveling, and chaotic waves. It is also possible to regularise chaotic dynamics into traveling waves by enhancing the inertialess instabilities through the advective terms.

[1]National Science Foundation, Engineering and Physical Sciences Research Council (UK)

8:26AM A9.00003 Instabilities and nonlinear waves in two-layer film flowing down a vertical plane , GOKCEN CEKIC, GRIGORY SISOEV, University of Birmingham, UK — The two-layer falling film flowing down a vertical plane is considered. The approximate long-wave model is investigated and the integral method is applied on this model. The linear stability of the steady flow is analyzed by the numerical method. To calculate the steady-traveling nonlinear waves we reformulate the problem as the dynamical system and find the bifurcating solutions. Examples of solutions at real-life values of the similarity parameters for a two-layer film are shown.

8:39AM A9.00004 Coherent structures in non-local active-dissipative equations , TE-SHENG LIN, Department of Mathematical Sciences, Loughborough University, MARC PRADAS, SERAFIM KALLIADASIS, Department of Chemical Engineering, Imperial College London, DEMETRIOS PAPAGEORGIOU, Department of Mathematics, Imperial College London, DMITRI TSELUIKO, Department of Mathematical Sciences, Loughborough University — We investigate a non-local weakly nonlinear equation arising in the modeling of wave dynamics on electrified falling films. The equation is a generalized Kuramoto-Sivashinsky (gKS) equation with a non-local term representing the imposed electric field. As for the case of the usual gKS equation, we find that sufficiently strong dispersion arrests the spatio-temporal chaos and the solutions evolve into arrays of pulses, each one of which resembles an infinite-domain pulse. Such pulses interact with each other and may form bound states. The Shilnikov-type approach for analyzing bound states is not applicable to non-local equations. We therefore develop an accurate weakly interaction theory for the pulses that allows us to analyze the attraction and repulsion of the pulses and the existence of bound states. The non-locality of the equation results in the fact that the infinite-domain pulse has algebraically decaying tails (in contrast to exponentially decaying tails for the local equation), which has strong effect on the interaction of the pulses. We compare the interaction theory with numerical simulations of the full equation and find very good agreement.

8:52AM A9.00005 Linear and nonlinear instability and ligament dynamics in 3D laminar two-layer liquid/liquid flows[1] , LENNON Ó NÁRAIGH, University College Dublin, PRASHANT VALLURI, The University of Edinburgh, DAVID SCOTT, IAIN BETHUNE, Edinburgh Parallel Computing Centre, The University of Edinburgh, PETER SPELT, Département Mécanique, Université de Lyon 1 and Laboratoire de Mécanique des Fluides & d'Acoustique (LMFA), CNRS, Ecole Centrale Lyon — We consider the linear and nonlinear stability of two-phase density-matched but viscosity contrasted fluids subject to laminar Poiseuille flow in a channel, paying particular attention to the formation of three-dimensional waves. The Orr–Sommerfeld–Squire analysis is used along with DNS of the 3D two-phase Navier–Stokes equations using our newly launched TPLS Solver (http://edin.ac/10cRKzS). For the parameter regimes considered, we demonstrate the existence of two distinct mechanisms whereby 3D waves enter the system, and dominate at late time. There exists a direct route, whereby 3D waves are amplified by the standard linear mechanism; for certain parameter classes, such waves grow at a rate less than but comparable to that of most-dangerous two-dimensional mode. Additionally, there is a weakly nonlinear route, whereby a purely spanwise wave couples to a streamwise mode and grows exponentially. We demonstrate these mechanisms in isolation and in concert. Consideration is also given to the ultimate state of these waves: persistent three-dimensional nonlinear waves are stretched and distorted by the base flow, thereby producing regimes of ligaments, "sheets," or "interfacial turbulence."

[1]HECToR RAP/dCSE Project e174, HPC-Europa 2

9:05AM A9.00006 Contact line instability of gravity-driven flow of power-law fluids: Comparison of Experiments and Simulations[1] , BIN HU, HENRY CLEVER, SARAH KIEWEG, University of Kansas — We previously studied the fingering instabilities of power-law fluids using linear stability analysis (LSA). We also developed a 3D FEM model to simulate a constant-volume power-law fluid flowing down an incline. In this study, we try to perform 3D simulations with constant-flux condition and perturbed contact line, and compare the results to LSA. Moreover, we develop a fluid depth measurement experiment based on fluorescence imaging for further comparison to the numerical results. Instead of using laser-induced fluorescence, we try a simple quantitative way of using LEDs, which is much less expensive. The impact of inclination angle, surface tension, and especially shear-thinning effect on contact line instabilities is investigated.

[1]This work is supported by NIH R21/R33 Microbicide Innovation Program.

9:18AM A9.00007 Spin coating flow of Power law fluid: spreading and contact line intsability[1] , PANKAJ DOSHI, National Chemical Laboratory, Pune, India, AKASH ARORA, Department of Chemical Engineering and Material Science, University of Minnesota — A computational study of the flow of a power law fluid on a spinning disc is considered here. The main goal of this work is to examine the effect of shear thinning nature on the flow development and associated contact line instability. The governing mass and momentum balance equations are simplified using the lubrication theory. The resulting model equation is a fourth order non-linear PDE which describes the spatial and temporal evolution of film thickness. The movement of contact line is modeled using a constant angle slip model. To solve this moving boundary problem, a numerical method is developed using a Galerkin finite element method (G/FEM) based approach. The numerical results show that the spreading rate of the fluid increases with the increase in the shear-thinning character of the fluid. It is also observed that the sharpness of capillary ridge is reduced as the shear-thinning character of the fluid becomes dominant. In order to study the stability of these ridges, linear stability theory is developed for shear thinning fluid. The dispersion relationship depicting the growth rate for a given wave number have been reported and compared for different power-law fluids. It is found that the growth rate of the fingers decreases as the fluid becomes more shear-thinning in

[1]Department of Science and Technology, India

9:31AM A9.00008 Engineering and control of surfactant-laden flows: experiments and MD simulations[1] , NINA KOVALCHUK, University of Loughborough, PANAYIOTIS THEODORAKIS, ERICH MULLER, RICHARD CRASTER, Imperial College London, VICTOR STAROV, University of Loughborough, OMAR MATAR, Imperial College London — The dynamics of surfactant-laden flows remain full of surprises. For hydrophobic substrates with a water contact angle of less than 110°, certain types of surfactants, known as *superspreaders*, can lead to an increase in the spreading factor by two orders of magnitude over water droplets; spreading takes place with speeds between 1-10 mm/s. The superspreading effect occurs provided the concentration of superspreaders is above the critical wetting concentration (CWC), which, in turn, exceeds (by several times) the critical aggregation concentration. The CWC is dependent on the type of surfactant but independent of the nature of the substrate. In this study, we use a combination of molecular dynamics simulation, and direct experimentation to analyse the spreading behaviour of well-known superspreaders. We correlate this behaviour in terms of the physic-chemical properties of the surfactant (sorption kinetics, aggregation formation, and dynamic surface tension).

[1]EPSRC Grant (EP/J010502/1)

9:44AM A9.00009 New spreading law of thin film liquids controlled by gravity and vdW forces under thermal fluctuations , SVETOZAR NESIC[1], RODOLFO CUERNO REJADO[2], ESTEBAN MORO EGIDO[3], Universidad Carlos III de Madrid — It has been shown that, in the regime controlled by surface tension, the spreading dynamics of a thin viscous fluid droplet changes significantly when it is subjected to thermal fluctuations. Technically, this has been accomplished through the incorporation of appropriate stochastic terms into the standard lubrication equation. In practice, it leads to a modification of the classic Tanner's law for spreading, with implications for Micro and Nanofluidic systems. We have recently found a new law of spreading for the same kind of systems, but in the gravity-dominated regime. Moreover, in the deteministic case a finite contact angle is formed when a van der Waals attractive force is introduced to the system and we show that there is a slight change in contact angle when thermal fluctuations are taken into account.

[1]Ph.D student and a member of GISC (http://matematicas.uc3m.es/index.php/gisc)
[2]Associate Professor and a member of GISC
[3]Associate Professor and a member of GISC

Sunday, November 24, 2013 8:00AM - 9:31AM –

Session A10 Jets I: Swirling, Mixing and Multiphase 334 - Kenneth T. Kiger, University of Maryland

8:00AM A10.00001 Stability of swirling coaxial jets[1] , JESSIE WELLER-CALVO, LAURENT JOLY, JEROME FONTANE, Universite de Toulouse, ISAE — In order to improve the mixing properties of injectors, we investigate the potential synergy between azimuthal and axial shear. To this end, we examine the linear modal stability of a simplified analytical model which consists of a temporally evolving swirling jet surrounded by an annular jet with a different axial velocity. We denote $\Lambda = V_2/V_1$ the ratio between the axial velocity of the non-swirling outer jet V_2 and the axial velocity of the central jet V_1; and $q = \Omega_c r_1/V_1$ the swirl number of the central jet where Ω_c is the rotation rate on the axis and r_1 the central jet radius. The present study extends the results of Gallaire & Chomaz (2003) where a single swirling jet was considered. For all values of the swirl number up to $q = 2$, adding the outer non-swirling jet substantially increases the growth rate of the most amplified mode, which can be more than doubled when $\Lambda > 1$. This is the result of the collaborative axial and azimuthal shear instabilities localised in between the two jets. The mode selection of larger azimuthal wavenumbers with increasing q, identified by Gallaire & Chomaz, is no longer observed when the outer jet is at least as fast as the central jet $\Lambda > 1$, the axisymmetric mode being the most amplified.

[1]Founded by DGA MRIS

8:13AM A10.00002 Axial Evolution of Helical Modes in Reacting Swirl Flow , MIKE AGUILAR, BENJAMIN EMERSON, DAVID NOBLE, TIM LIEUWEN, Georgia Institute of Technology — The swirling jet is a common method of flame stabilization for ground power and aviation combustors. The hydrodynamic stability characteristics of high Reynolds number, reacting, swirling jets are not well understood, but have important influences on the flame stability and combustion dynamics of these combustors. These systems exhibit a variety of unsteady motions associated with the globally unstable vortex breakdown region, as well as the convectively unstable shear layers. Depending upon whether the system is executing natural oscillations or is externally forced at some other frequency (such as during a combustion instability), the dominant shear layer modes can vary between symmetric, m=0 structures, to various helical m = +1 or +2 structures. In addition, these modes evolve axially in different ways. This study presents 10 kHz PIV measurements of such a reacting swirl flow, with and without harmonic forcing. Modal decomposition is used on the measured velocity fields to extract the dominant mode shape and its frequency and spatial growth. Next, hydrodynamic stability theory is used on the measured time averaged flow field to predict the dominant vortex shedding mode and its axial growthrate. The predictions are compared to the experimental results for various flow conditions and forcing arrangements. Finally, we comment on the utility of linear stability methods for predicting the dominant mode and its spatial growth in reacting, swirling jets.

8:26AM A10.00003 The coiling of electrified liquid jets[1] , JAVIER RIVERO RODRIGUEZ, MIGUEL PÉREZ-SABORID, Universidad de Sevilla — We have carried out a numerical study of the coiling regime which takes place when an electrified liquid jet issuing from an orifice drilled in a metal plate electrode reaches the counter electrode. Based on the slenderness assumption, we have derived the set of one-dimensional dynamical equations by averaging the underlying balance laws over the jet cross sections (Cosserat rod model). Therefore, our equations and boundary conditions are related to those obtained by N.M. Ribe (Ann. Rev Fluid Mech., 2012) for the coiling of liquid ropes, but including electrostatic effects. In a first approach, we have simplified the electrical terms entering the problem by assuming a constant external electric field between electrodes, and that the charges are convected by the jet surface interacting electrostatically with each other via the local interaction approximation (Yarin et al., 2001). We have numerically investigated the problem in order to analize how the coiling regime depends on the dimensionless parameters of the problem, i.e., the Reynolds number, the electrical Bond number and the capillary number. In particular, we have found that both the displacement of the centerline of the jet and its cross-sectional stretching greatly depend on the electrostatic effects.

[1]Thanks to the MINECO (Spain) for partial support under grant DPI2010-20450-C03-02

8:39AM A10.00004 Study of the turbulent diffusive flux for tracer dispersion in quasi-two-dimensional turbulent jets , JULIEN R. LANDEL, DIMITRY FOURES, DAMTP, University of Cambridge, C.P. CAULFIELD, BPI & DAMTP, University of Cambridge — The study of turbulent jets in relatively enclosed geometries is relevant to rivers flowing into lakes. In the event of a spillage of pollutants into a river, it is critical to understand how these agents disperse with the flow in order to assess damage to the environment. Using the ensemble-averaged time-dependent advection-diffusion equation, we obtain an equation for the temporal and spatial evolution of the ensemble-averaged concentration of passive tracers injected in a steady turbulent quasi-two-dimensional jet. The turbulent flux of the concentration fluctuations is responsible for the lateral and streamwise dispersion in the jet. We reconstruct this second-order turbulent flux using two independent experimental measurements of the concentration field and of the time-averaged velocity field. We present results in the case of constant-flux releases of tracers in the jets. We find that the dispersion properties differ significantly between the lateral and streamwise direction. Due to a large streamwise dispersion a significant amount of tracers can be transported faster than the speed predicted by a simple top-hat advection model in the jet.

8:52AM A10.00005 Optimum viscous flow in pressure-swirl atomizers , GHOBAD AMINI, AARON PEREIRA, University of Waterloo, SANGSIG YUN, National Research Council of Canada, XIANGUO LI, University of Waterloo — Due to their simple configuration and reliable operation, pressure-swirl atomizers are widely used in applications such as combustion, painting, humidification, and sprinkling. The liquid is swirled by entering into the atomizer tangentially and its surface area is increased as discharges in a large spray angle. Understanding the effects of nozzle geometry and inlet flow condition on the discharge coefficient and spray angle is very important in nozzle design. To this end, the flow field inside a pressure-swirl atomizer has been studied theoretically. The main body of the liquid is taken to be moving in circles round the axis. Within the boundary layer, containing transverse and longitudinal velocity components, the retarded liquid is slowed down by viscosity and driven towards the exit orifice by pressure gradient. The swirling motion of liquid creates a low pressure zone near the nozzle axis and leads to the formation of a helical air-core. Through studying the growth of the boundary layer from nozzle entry to the orifice exit, the portions of the outflow exits the orifice from boundary layer current and also from the main body of the swirling liquid are specified. For a given range of pressure drop values, the optimum nozzle geometry and liquid flowrate are predicted. Additionally, the reason of increasing the flow by increasing liquid viscosity or decreasing orifice diameter is explained. A series of experiments and numerical modeling have also been carried out to support the theoretical results.

9:05AM A10.00006 Structure, Mixing, and Stability of Flush and Elevated Jets in Crossflow[1] , LEVON GEVORKYAN, DANIEL GETSINGER, TERRY WEN YU PENG, OWEN SMITH, ANN KARAGOZIAN, University of California, Los Angeles — The present experiments explore the characteristics of equidensity and variable density transverse jets using acetone PLIF, stereo PIV, and hot wire anemometry. Jets composed of mixtures of helium and nitrogen are injected normally from different types of nozzles (flush and elevated with respect to the wind tunnel wall, and converging as well as straight shapes) into an air crossflow. A range of jet-to-crossflow momentum flux ratios J and density ratios S is examined, within which previous studies[2] have identified conditions for upstream shear layer transition from convective to absolute instability. The present study examines the relationships among transverse jet structure, including vortical rollup and cross-sectional symmetry/asymmetry, jet mixing characteristics, and shear layer stability characteristics. The role of the crossflow boundary layer as well as jet injection systems for structure, mixing, and stability is evaluated and related to prior observations on vorticity evolution for jets in crossflow.

[1]Supported by NSF grant CBET-1133015 & AFOSR grant FA9550-11-1-0128 (A001768901)

[2]Megerian, et al., **JFM**, 2007; Davitian, et al., **JFM**, 2010; Getsinger, et al., **Expts in Fluids**, 2012

9:18AM A10.00007 Dynamics of viscous fluid jets containing solid particles at low Reynolds number[1] , MICHAEL NORTON, TERESA BRUGAROLAS, JONATHAN CHOU, DAEYEON LEE, HAIM BAU, University of Pennsylvania — Using a high-speed camera and a glass capillary flow focusing device, we observe the effects of suspended, elongated particles (aspect ratios 1 - 10) on the dynamics of low Reynolds number water jets ejected into oil (containing surfactant) and the size distribution of the droplets resulting from the jet breakdown. We report on the interaction between the jets and the particles in both the absolutely unstable (dripping) regime and the convectively unstable (jetting) regime. In the former, particles induced coalescence of droplets. In the jetting mode, in addition to coalescence, the presence of particles in the jet caused variations in droplets' sizes both upstream and downstream of the droplet that houses a particle. In the jetting regime, particles circulating in the cone of the jet upstream of the nozzle excited periodic disturbances in the jet that induced variations in droplet sizes.

[1]NIH 1R03AG042690 - 01A1

Sunday, November 24, 2013 8:00AM - 9:57AM –

Session A11 Bubbles I: Cavitation, Nucleation and Ventilation 335 - Tim Colonius, California Institute of Technology

8:00AM A11.00001 Pressures induced by collapsing cavitation bubbles near boundaries , JOEL HARTENBERGER, RENAUD GAUDRON, ERIC JOHNSEN, STEVEN CECCIO, University of Michigan, Ann Arbor — A pulsed Nd:YAG laser was used to produce single cavitation bubbles both in the free field and near solid and compliant boundaries in a quiescent flow. High-speed videography was used to record the growth and collapse of these bubbles. Simultaneously, the local impulse created by the bubbles was measured using a needle hydrophone. The goal of the study is to determine how impulses are created during both the initial collapse of the bubble during re-entrant jetting flow and the final collapse of the bubble after jet impact. The needle probe was used to spatially and temporally resolve the creation of force normal to the surface of the boundary. Results will be used to validate numerical simulations of single cavitation bubble collapse testing several bubble dynamics models.

8:13AM A11.00002 Inertial cavitation threshold in nonlinear viscoelastic media , MATTHEW WARNEZ, ERIC JOHNSEN, University of Michigan — Thresholds for inertial cavitation in tissue are studied through spherical bubble models coupled to viscoelastic constitutive relationships. Therapeutic ultrasound treatments aim to exploit the large strains and shockwaves caused by large-amplitude bubble oscillations, but metrics for the onset of inertial cavitation in soft tissue do not readily carry over from water-based cavitation. Tissue is represented by a Zener-type model that incorporates viscosity, neo-Hookean elasticity, and upper-convected Maxwell relaxation. The partial differential equations for stress are solved via a spectral collocation method. The bubble dynamics are described by the Keller-Miksis equation with thermal effects. New metrics for viscoelastic cavitation thresholds are proposed and compared against past metrics. The influence of viscoelastic parameters and choice of constitutive relationship on bubble behavior is investigated in detail.

8:26AM A11.00003 Cavitation dynamics in a viscoelastic medium with nonlinear elasticity , RENAUD GAUDRON, ERIC JOHNSEN, University of Michigan — Past methods for modeling the dynamics of a spherical cavitation bubble in a viscoelastic medium (e.g., soft tissue) usually assume the elasticity to be linear. In this work, we develop a general framework to study cavitation in nonlinear (visco)elastic media, which are expected to be more accurate for large-amplitude bubble oscillations. By following an approach based on deformation tensors and Cauchy stresses, the models presented here not only take into account the usual viscous, inertial, pressure and surface tension effects, but also complex nonlinear elasticity directly derived from specific strain-energy functions. The present results are consistent with past studies of linear viscoelasticity, but additional elastic terms with different exponents emerge in the bubble dynamics equation (e.g., Rayleigh-Plesset) for more complicated strain-energy functions. Key quantities in cavitation dynamics (bubble natural frequency, minimum radius, etc.) are reported for the neo-Hookean model, the simplest nonlinear elastic model. This approach also readily leads to a full description of the physical variables of the medium where the bubble oscillates (pressure, strain/strain rate, stress, etc.).

8:39AM A11.00004 Detailed Simulations of Bubble-Cluster Collapse Adjacent Material Surfaces , ARPIT TIWARI, CARLOS PANTANO, JONATHAN B. FREUND, University of Illinois at Urbana-Champaign — The collapse of bubble clusters adjacent material surfaces is thought to be an important damage mechanism, in both engineering and biomedical applications. Homogeneous models of these clusters have been able to reproduce some of their gross dynamics, however diagnostic challenges leave it unclear how important the bubble dynamics are for important quantities such as peak pressures on the surface. We study in detail the dynamics of small clusters collapsing adjacent to a wall using a numerical scheme that faithfully represents bubble-scale dynamics. It is based on a recently developed interface capturing method that is asymptotically consistent with a well-posed mixture model for the two phases. For collapse near a rigid wall, we show strong inward focusing of re-entrant jets, which enhances the impulsive pressures generated on the wall. The homogeneous model we compare with fails to capture the true peak pressures on the walls. We further apply our scheme to simulate cluster collapse near a viscous fluid as a model for soft tissue, as in therapeutic ultrasound. In this case, the low impedance mismatch at the wall leads to significantly different dynamics. Simulations suggest that clusters can actually be relatively protective when compared to single-bubble collapses.

8:52AM A11.00005 Shear stresses and temperatures during the collapse of a bubble near a rigid wall[1] , SHAHABODDIN ALAHYARI BEIG, ERIC JOHNSEN, Department of Mechanical Engineering, University of Michigan, Ann Arbor, United states — The collapse of a cavitation bubble is the central problem in cavitation erosion. In naval and biomedical applications, this process is known to lead to structural damage, whether intended or not. In the present work, the collapse of an initially spherical bubble, filled with non-condensable gas, near a rigid wall is simulated numerically using a high-order shock- and interface-capturing scheme. This computational approach prevents both temperature and pressure errors by using appropriate transport equations for the variables entering the equation of state. By directly solving the axisymmetric compressible Navier-Stokes equations, the viscous stresses and temperatures produced along the neighboring wall are computed. The quantities are critical when considering compliant bodies and polymeric coatings on metallic surfaces. The simulations show that substantial increases in temperature in the liquid may be produced. Characterization of the temperatures and viscous stresses along the neighboring wall will be presented.

[1]ONR grant N00014-12-1-0751

9:05AM A11.00006 Why does a beer bottle foam up after a sudden impact on its mouth? , JAVIER RODRIGUEZ-RODRIGUEZ, ALMUDENA CASADO-CHACON, Carlos III University of Madrid, Spain, DANIEL FUSTER, CNRS (UMR 7190), Universite Pierre et Marie Curie, Institut Jean le Rond d'Alembert, France — A sudden vertical impact on the mouth of a beer bottle generates a compression wave that propagates through the glass towards the bottom. When this wave reaches the base of the bottle, it is transmitted to the liquid as an expansion wave that travels to free surface, where it bounces back as a compression wave. This train of expansion-compression waves drives the forced cavitation of existing air pockets, leading to their violent collapse. A cloud of very small daughter bubbles are generated upon these collapses, that expand much faster than their mothers due to their smaller size. These rapidly growing bubble clusters effectively act as buoyancy sources, what leads to the formation of bubble-laden plumes whose void fraction increases quickly by several orders of magnitude, eventually turning most of the beverage into foam. In this talk, we will analyze quantitatively these processes in order to explain the extremely high efficiency of the degasification process that occurs in the bottle within the few seconds that follow the impact. This work has been supported by Spanish Ministries of Science and of Economy and Competitiveness through grants: DPI2008-06369 and DPI2011-28356-C03-02.

9:18AM A11.00007 Bubble cloud nucleation induced by the interaction between multiple laser-induced shocks and bubbles , PEDRO QUINTO-SU, Instituto de Ciencias Nucleares, Universidad Nacional Autónoma de México, KEITA ANDO, Department of Mechanical Engineering, Keio University — Multifocal laser-induced optical breakdown in water is used to nucleate microscopic bubble clouds. The liquid is ruptured via the interaction of multiple shocks and bubbles. We find that the liquid is fractured at localized regions defined by the planes bisecting each pair of foci on the array, where rarefaction waves (reflected from the laser bubbles) merge. For laser pulses focused at two spots we measure the probability for nucleation as a function of separation between the foci and the maximum tensions are calculated with Euler flow simulations.

9:31AM A11.00008 Ventilation of an hydrofoil wake[1] , ROGER ARNDT, Retired, SEUNG JAE LEE, GARRETT MONSON, Saint Anthony Falls Laboratory, University of Minnesota — Ventilation physics plays a role in a variety of important engineering applications. For example, hydroturbine ventilation is used for control of vibration and cavitation erosion and more recently for improving the dissolved oxygen content of the flow through the turbine. The latter technology has been the focus of an ongoing study involving the ventilation of an hydrofoil wake to determine the velocity and size distribution of bubbles in a bubbly wake. This was carried out by utilizing particle shadow velocimetry (PSV). This technique is a non-scattering approach that relies on direct in-line volume illumination by a pulsed source such as a light-emitting diode (LED). The data are compared with previous studies of ventilated flow. The theoretical results of Hinze suggest that a scaling relationship is possible that can lead to developing appropriate design parameters for a ventilation system.

[1]Sponsored by ONR and DOE

9:44AM A11.00009 Improved Performance With Ventilation[1] , ELLISON KAWAKAMI, 3 M Corporation, SEUNG JAE LEE, ASHISH KARN, JIARONG HONG, Saint Anthony Falls Laboratory, University of Minnesota, ROGER ARNDT, Retired — 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. Various aspects of the flow physics of a supercavitating vehicle have been under investigation for several years at Saint Anthony Falls Laboratory. Both steady flow and simulated flow below a wave train have been studied. Using a high speed camera and the proper software, it is possible to synchronize cavity dimensions with pressure measurements taken inside the cavity to permit an in-depth study of unsteadiness. It was found that flow unsteadiness caused a decrease in the overall length of the supercavity while having only a minimal effect on the maximum diameter. Results regarding supercavity shape, ventilation demand, cavitation parameters and closure methods are reviewed in light of new studies that focused on various closure mechanisms.

[1]Sponsored by ONR.

Sunday, November 24, 2013 8:00AM - 9:57AM –

Session A12 Vortex Dynamics and Vortex Flows I 336 - Joseph Bull, University of Michigan

8:00AM A12.00001 Vortical wake evolution and its effect on performance using Lagrangian coherent structures , TIMOTHY JETER, MELISSA GREEN, Syracuse University — In the field of bio-inspired hydrodynamics, positive thrust producing wakes and their evolution are of particular interest. Water tunnel experiments that utilize a vertically-mounted low-aspect-ratio flat panel are actuated in a purely pitching motion by employing a two-axis motion controller. Vortical wake structure data are collected using stereo particle image velocimetry (SPIV), and the velocity fields are analyzed using the Eulerian Q criterion and the Lagrangian finite time Lyapunov exponent (FTLE). We validate specific assumptions and results of previous work done with a similar geometry such as a negligible spanwise velocity at the midspan of the wake, and a strong spanwise induced velocity near the edges of the wake. The stereo analysis provides a quantitative measurement of the spanwise velocity at selected locations to determine how important three-dimensional effects are and where they are originating.

8:13AM A12.00002 Zero-Net Mass-Flux Actuator Cavity Vortex[1] , MICHAEL KRIEG, KAMRAN MOHSENI, University of Florida — Zero-Net Mass-Flux (ZNMT) devices are used commonly as synthetic jet actuators for flow control in various applications. The authors have recently proposed using larger ZNMF jet actuators for underwater propulsion; similar to squid and jellyfish. Generally the external flow generated by these devices is characterized according to momentum and energy transfer rates, and little attention is paid to the dynamics of flow inside the cavity. In fact the flow inside the cavity, especially during the refilling phase is not only highly dynamic but greatly influences the pressure distribution at the opening as well as the external flow during the following jetting phase. A completely transparent axisymmetric ZNMF cavity was constructed in order to investigate the internal vortex dynamics. The flow is seeded with reflective particles and illumined with a laser sheet bisecting the axis of symmetry. Standard 2D DPIV techniques are used to recover the velocity field in this cross section. During filling it is observed that a starting jet extending from the opening to the inside of the cavity rolls into a vortex ring much like the jetting phase. However, the effect of the cavity walls becomes apparent almost immediately. In this talk we characterize how the circulation within the cavity decays as a function of both cavity/orifice geometry and the mass flux program. In addition a load cell measures the total thrust acting on the device which is used to validate pressure calculations performed on the moving surface inside the cavity, showing excellent agreement.

[1]This work is supported by a grant from the Office of Naval Research

8:26AM A12.00003 Evolution of the pressure thrust in a starting jet , LEI GAO, Nanyang Technological University, SIMON C.M. YU, Singapore Institute of Technology, JORG SCHLUTER, Nanyang Technological University — It is known that the nozzle exit over-pressure is responsible for the improved propulsive performance of a starting jet. To illustrate the detailed evolution of the pressure thrust during the vortex ring formation process, starting jets with a straight nozzle configuration are investigated numerically for different velocity programs. It is found that the remarkable over-pressure contribution to the unsteady jet thrust is mainly associated with the initial acceleration phase of the starting flow. If the vortex ring does not gain appreciable translational velocity at the end of the acceleration phase, it will induce a locally lower pressure region near the nozzle edge due to a mechanism similar to that for the leading edge suction force on a delta wing. As a result of the lower pressure at the nozzle exit plane, the pressure thrust contributes adversely to the total jet thrust. This negative pressure thrust diminishes rapidly as the leading vortex ring translates downstream away from the nozzle exit. Finally, after the leading vortex ring pinches off from the trailing jet, its effect on the pressure variation at the nozzle exit plane becomes negligible and the propulsive characteristics of the flow approaches that of a steady jet.

8:39AM A12.00004 Impact of Cyclical Pulse Behavior on Toroidal Vortex Interaction[1] , LOUIS SALMON, Louisiana Tech University, JOHN BAKER, The University of Alabama — Vortex rings formed by impulsively started jets have been of great interest due to the possible applications to underwater vehicle propulsion and as a means of inducing fluid flow. So far, most research into vortex rings has only investigated the shedding of a single vortex ring at different formation numbers. This study investigated the behavior of vortex rings and the effects on the circulation when two vortex rings were shed one after the other from the same impulsively started jet. The computational fluid dynamics software ANSYS Fluent was used to perform this study. The geometry for the study was that of a pipe exiting into a volume of quiescent fluid and the impulsively started jet was modeled as a square wave velocity profile at the inlet of the pipe. The computational model was validated for a single vortex ring by comparing circulation data obtained to that found in previously published research. Once the model was validated, an analysis of vortex ring interaction was performed. This study considered formation numbers in the range of 1 to 4. The results showed that the second vortex ring either did not form or merged with the first vortex ring depending on the conditions. The combined vortex ring was found to have an increased circulation and an elongated shape.

[1]Funding form NSF REU Grant #1062611 is gratefully appreciated.

8:52AM A12.00005 Nested contour-dynamic models for axisymmetric vortex rings and vortex wakes , CLARA O'FARRELL, JOHN O. DABIRI, California Institute of Technology — Jetting swimmers, such as squid and jellyfish, propel themselves by forming vortex rings. It is known that vortex rings cannot grow indefinitely, but rather "pinch off" once they reach their physical limit, and that a decrease in efficiency of fluid transport is associated with pinch-off. Previously, the Norbury family of vortices has been used as a model for axisymmetric vortex rings, and the response of this family to shape perturbations has been characterized. We improve upon the Norbury models, using nested patches of vorticity to construct a family of models for vortex rings generated by a piston-cylinder apparatus at different stroke ratios. The perturbation response of this family is considered by the introduction of a small region of vorticity at the rear of the vortex, which mimics the addition of circulation to a growing vortex ring by a feeding shear layer. Model vortex rings are found to either accept the additional circulation or shed it into a tail, depending on the perturbation size. A change in the behavior of the model vortex rings is identified at a stroke ratio of three. We hypothesize that this change in response is analogous to pinch-off, and that pinch-off might be understood and predicted based on the perturbation responses of model vortex rings.

9:05AM A12.00006 Optimal propulsive efficiency of vortex enhanced propulsion[1] , ROBERT WHITTLESEY, Exponent, Inc., JOHN DABIRI, California Institute of Technology — The formation of coherent vortex rings in the jet wake of a vehicle has been shown to increase the propulsive efficiency of self-propelled vehicles. However, the effect of varying vortex ring formation characteristics has not been explored for vehicles at Reynolds numbers comparable to autonomous or manned submersible vehicles. In this work, we considered a range of vortex ring formation characteristics and found a peak in the propulsive efficiency where the vortex rings generated are coincident with the onset of vortex ring pinch off. This peak corresponds to a 22% increase in the propulsive efficiency for the vortex-enhanced wake compared to a steady jet.

[1]We gratefully acknowledge the support of the Office of Naval Research Grants N000140810918 and N000141010137.

9:18AM A12.00007 Propulsion by active and passive airfoil oscillation , A.W. MACKOWSKI, C.H.K. WILLIAMSON, Cornell University — Oscillating airfoils have been the subject of much research both as a mechanism of propulsion in engineering devices as well as a model of understanding how fish, birds, and insects produce thrust and maneuvering forces. Additionally, the jet or wake generated by an oscillating airfoil exhibits a multitude of vortex patterns, which are an interesting study in their own right. We present PIV measurements of the vortex flow behind an airfoil undergoing controlled pitching oscillations at moderate Reynolds number. As a method of propulsion, oscillating foils have been found to be capable performers when undergoing both pitching and heaving motions [Anderson et al. 1998]. While an airfoil undergoing only pitching motion is a relatively inefficient propulsor, we examine the effect of adding passive dynamics to the system: for example, actuated pitching with a passive spring in the heave direction. Practically speaking, a mechanical system with such an arrangement has the potential to reduce the cost and complexity of an oscillating airfoil propulsor. To study an airfoil undergoing both active and passive motion, we employ our "cyber-physical fluid dynamics" technique [Mackowski & Williamson, 2011] to simulate the effects of passive dynamics in a physical experiment.

9:31AM A12.00008 Formation of vortex pairs with hinged rigid flaps at the nozzle exit , PRASHANT DAS, RAGHURAMAN GOVARDHAN, JAYWANT ARAKERI, Indian Institute of Science — Biological flows related to aquatic propulsion using pulsed jets, or flow through the valves in a human heart, have received considerable attention in the last two decades. Both these flows are associated with starting jets that occur through biological tissue/membranes that are flexible. Motivated by these flows, we explore in the present work, the effect of passive flexibility of the nozzle exit on vortex generation from a starting jet. The starting jet is generated using a two-dimensional piston cylinder mechanism, the cross-section of the cylinder being rectangular with large aspect ratio. The fluid is pushed out of this cylinder or channel using a computer controlled piston. We introduce flexibility at the channel exit by hinging rigid flaps, which are initially parallel to the channel. The hinge used is such that it provides negligible stiffness or damping, thus allowing for the maximum opening of the flaps due to fluid forces. Using this system, we study both the flap kinematics and the vorticity dynamics downstream of the channel exit. Visualizations show large flap motions as the piston starts and this dramatically changes the vorticity distribution downstream of the flaps, with the formation of up to three different kinds of vortex pairs. This idealized configuration opens new opportunities to look at the effect of flexibility in such biological flows.

9:44AM A12.00009 The Formation of Turbulent Vortex Rings by Synthetic Jets , JOHN LAWSON, University of Cambridge, JAMES DAWSON, The Norwegian University of Science & Technology — Vortex rings formed by synthetic jets are found in many engineering and biological flows. For vortex rings formed both periodically and in isolation, a constraint on vortex formation ("pinch-off") has been observed which is relevant to unsteady propulsion. However, there is no clear consensus on the physical mechanism of this constraint. We present analysis of time resolved, 2D Particle Image Velocimetry measurements of the velocity and material acceleration field in an axisymmetric, turbulent synthetic jet in air at maximum stroke ratios $L_m/D = 2 - 15$. Using the acceleration field, pinch-off may be identified in a manner which is frame invariant and consistent with previous studies. An adverse pressure gradient behind the ring and induced by it plays a role in the pinch-off and separation of the ring from the jet. Recognising this, we revise an existing model for pinch-off: this revision fits our data well. Additionally, we show that as the ring forms, hydrodynamic impulse is delivered via two equally important mechanisms: a material flux and a vortex force. For large L_m/D, this vortex force may deliver a substantial impulse to the ring after pinch-off. This has implications for unsteady propulsion, models of vortex ring formation and existing explanations for pinch-off.

Sunday, November 24, 2013 8:00AM - 9:57AM –

Session A13 Focus Session: Marine Hydrokinetic Energy Conversion I 301 - Laura Beninati, Bucknell University

8:00AM A13.00001 Interaction between an axial-flow model hydrokinetic turbine and an erodible channel , CRAIG HILL, University of Minnesota - St. Anthony Falls Laboratory, MIRKO MUSA, University of Trento, LEONARDO P. CHAMORRO, University of Illinois - Department of Mechanical Science and Engineering, MICHELE GUALA, University of Minnesota - St. Anthony Falls Laboratory — Laboratory experiments were carried out to examine the effect of relatively large-scale bedforms on the performance of a model axial-flow hydrokinetic turbine. The turbine rotor, $d_T = 0.15$m, was attached to a miniature DC motor, and allowed for voltage data acquisition at 200 Hz along with 3D hub-height inflow velocity, U_{hub}, approximately $7d_T$ upstream of the turbine. Spatio-temporal bed elevations were acquired along three longitudinal sections and at least one transverse transect within the flume providing the temporally-averaged scour and deposition patterns characterizing the turbine near-field region. Turbine-turbine interaction was investigated under aligned configurations in the streamwise direction with variable spacing both in clear water scour and live bed transport conditions. Effects from both migrating bedforms and the upstream turbine were observed in the long-term and short-term voltage fluctuations of the downstream turbine. Combined measurements of inflow velocity, bed topography and turbine voltage were used to obtain joint statistics and correlations, which provided an indication of the variability in environmental exposure and performance that hydrokinetic turbines will encounter in natural erodible rivers.

8:13AM A13.00002 Tow tank measurements of turbulent flow in the near wake of a horizontal axis marine current turbine under steady and unsteady inflow conditions , LUKSA LUZNIK, MAX VAN BENTHEM, KAREN FLACK, ETHAN LUST, US Naval Academy — Near wake measurements are presented for a 0.8 m diameter (D) two bladed horizontal axis tidal turbine model for two inflow conditions. The first case had steady inflow conditions, i.e. turbine was towed at a constant carriage speed and the second case had a constant carriage speed and incoming regular waves with a period of 1.6 seconds and 0.09 m wave height. The test matrix in the wake covered four radial positions from r/D=0.3 to 0.5 and five axial positions from x/D=0.19 to 0.95. All measurements were performed at the nominal tip speed ratio (TSR) of 7.4. The distribution of mean velocities for the steady inflow case exhibit significant spatial variability in the wake region. Normalized mean streamwise velocity show a decrease in magnitude with the axial direction for all radial locations ranging from U/Utow=0.55 at r/D=0.49 to 0.35 at r/D=0.3. Vertical and lateral mean velocities are small but consistent with counterclockwise fluid angular momentum for a clockwise rotor rotation. The Reynolds shear stresses consistently show elevated levels for measurements near the rotor tip (r/D=0.49) and are significantly reduced by x/D=0.6 downstream. This suggests low turbulence levels in the wake which is consistent with very low free stream turbulence. For the case with waves, evidence of enhanced turbulence intensities and shear stresses within spatial coverage of the experiment suggest increased in localized turbulence production in the blade tip region over the entire near wake region.

8:26AM A13.00003 An investigation into blockage corrections for cross-flow hydrokinetic turbine performance , ROBERT CAVAGNARO, BRIAN POLAGYE, University of Washington — The performance of hydrokinetic turbines is augmented in confined channels, such that the coefficient of performance is elevated versus free-stream conditions. This often introduces uncertainty when characterizing prototype-scale turbines in flume or tow tank facilities. Performance of a one-quarter scale helical, cross-flow turbine is characterized over a range of operating conditions (inflow velocity and tip-speed ratio) at blockage ratios (ratio of rotor swept area to channel area) of ∼10 and ∼25%. Particle image velocimitry is used to characterize rotor induction, as well as the turbulent wake produced by the turbine. Performance at the different blockage ratios is compared to corrections derived from actuator disk theory and to full-scale field performance in the absence of blockage.

8:39AM A13.00004 Performance and cavitation characteristics of bi-directional hydrofoils , IVAYLO NEDYALKOV, MARTIN WOSNIK, Center for Ocean Renewable Energy, University of New Hampshire — Tidal turbines extract energy from flows which reverse direction. One way to address this bi-directionality in horizontal axis turbines that avoid the use of complex and maintenance-intensive yaw or blade pitch mechanisms, is to design bi-directional blades which perform (equally) well in either flow direction. A large number of proposed hydrofoil designs were investigated using numerical simulations. Selected candidate foils were also tested (at various speeds and angles of attack) in the High-Speed Cavitation Tunnel (HiCaT) at the University of New Hampshire. Lift and drag were measured using a force balance, and cavitation inception and desinence were recorded. Experimental and numerical results were compared, and the foils were compared to each other and to reference foils. Bi-directional hydrofoils may provide a feasible solution to the problem of reversing flow direction, when their performance and cavitation characteristics are comparable to those for unidirectional foils, and the penalty in decreased energy production is outweighed by the cost reduction due to lower complexity and respectively lower installation and maintenance costs.

8:52AM A13.00005 Experimental characterization of marine hydrokinetic (MHK) turbine array performance , NICKOLAS STELZENMULLER, ALBERTO ALISEDA, University of Washington — Three scale model horizontal axis MHK turbines (1:45) were tested in a flume at various array spacings. The scale rotors are based on the full-scale Department of Energy Reference Model 1, modified to reproduce the hydrodynamic performance of the full-scale turbine at the reduced experimental Reynolds number (10^5 vs 10^6, based on chord length). Flow incident on the turbines and in the wakes was characterized via PIV and ADV measurements. Tip speed ratio (TSR) similarity of the turbines is achieved by controlling the applied load with magnetic particle brakes. Single turbines were characterized at various mean freestream velocities to explore the effect of Reynolds number on turbine performance. Measured turbine efficiencies of ∼40% are similar to efficiencies predicted from full-scale simulations, indicating similar energy extraction at scale. Wake characteristics and turbine efficiencies have been investigated at a range of TSRs, with the goal of determining array spacing and operating conditions that maximize overall array efficiency. Free surface deformations were measured near the rotor plane for various vertical positions of the turbine relative to the free surface and compared to numerical simulation results.

9:05AM A13.00006 Reynolds number effects on the performance and near-wake of a cross-flow turbine[1] , PETER BACHANT, MARTIN WOSNIK, Center for Ocean Renewable Energy, University of New Hampshire — To design wind or marine hydrokinetic (MHK) turbine farms with high efficiency, interactions between turbine wakes must be accurately predicted. However, to date numerical models predicting detailed wake properties of cross-flow (or vertical-axis) turbines have been validated with experimental data taken at Reynolds numbers significantly lower than those of full scale devices, casting doubt on the models' accuracy. To address this uncertainty, we investigated the effects of Reynolds number on the performance and near-wake characteristics of a 3-bladed cross-flow turbine, both experimentally and numerically. Mechanical power output and overall streamwise drag were measured in a towing tank at turbine diameter Reynolds numbers $Re_D = 0.5 \times 10^5$–2.0×10^6. A detailed map of the near-wake one turbine diameter downstream was acquired via acoustic Doppler velocimetry for each Reynolds number case, from which differences in the mean velocity, turbulence intensity, and Reynolds stresses are highlighted. Finally, Reynolds-averaged Navier-Stokes (RANS) numerical simulations were performed, the results from which are compared with the experimental data.

[1]Work supported by NSF-CBET grant 1150797.

9:18AM A13.00007 Comparison of spatio-temporal resolution of different flow measurement techniques for marine renewable energy applications[1] , VINCENT LYON, MARTIN WOSNIK, Center for Ocean Renewable Energy, University of New Hampshire — Marine hydrokinetic (MHK) energy conversion devices are subject to a wide range of turbulent scales, either due to upstream bathymetry, obstacles and waves, or from wakes of upstream devices in array configurations. The commonly used, robust Acoustic Doppler Current Profilers (ADCP) are well suited for long term flow measurements in the marine environment, but are limited to low sampling rates due to their operational principle. The resulting temporal and spatial resolution is insufficient to measure all turbulence scales of interest to the device, e.g., "blade-scale turbulence." The present study systematically characterizes the spatial and temporal resolution of ADCP, Acoustic Doppler Velocimetry (ADV), and Particle Image Velocimetry (PIV). Measurements were conducted in a large cross section tow tank (3.7m x 2.4m) for several benchmark cases, including low and high turbulence intensity uniform flow as well as in the wake of a cylinder, to quantitatively investigate the flow scales which each of the instruments can resolve. The purpose of the study is to supply data for mathematical modeling to improve predictions from ADCP measurements, which can help lead to higher-fidelity energy resource assessment and more accurate device evaluation, including wake measurements.

[1]Supported by NSF-CBET grant 1150797.

9:31AM A13.00008 Upstream blockage effect on the thrust force of a marine hydrokinetic device , GIULIO SOLIANI, MARIA LAURA BENINATI, Bucknell University, MICHAEL KRANE, ARNOLD FONTAINE, Penn State University — The study evaluates the interaction of two model marine devices axially arranged one in front of the other, in a tandem configuration. Particular focus is given to the change that occurs in the thrust of the downstream marine hydrokinetic (MHK) device when the spatial arrangement of the two elements is varied. At critical spacing there is no thrust generation. The study is motivated by the need to predict the thrust behavior of MHK devices and determine the minimum separation distance to avoid the no thrust condition. The downstream element is a two-bladed, horizontal axis turbine, while the upstream blockage is a perforated disk with similar geometric properties intended to approximate the wake of the MHK device. Testing is conducted in the flume facility at Bucknell University. Experiments are performed for a fixed range of spacing between the perforated disk and the turbine. For each separation distance, the span-wise velocity profile upstream and downstream of the turbine is measured, as well as the device's rotational speed. The turbine's thrust coefficient is calculated. Plots of the thrust coefficient as a function of spacing depict the minimum separation distance to avoid the no thrust condition.

9:44AM A13.00009 Scour around a submerged cylinder and marine hydrokinetic (MHK) device in live-bed conditions , MARIA LAURA BENINATI, MICHAEL VOLPE, Bucknell University, MICHAEL KRANE, ARNOLD FONTAINE, Penn State University — Experiments are presented to explore how sediment scour around a single Marine Hydrokinetic (MHK) turbine varies with flow speed. Three Reynolds numbers, based on support structure diameter were used to induce live-bed scour conditions. Based on results from previous studies on submerged cylinders, differences in scour patterns between a single cylinder and MHK device can be determined. In the case of MHK energy, many devices are submerged in the flow. Thus, it is important to analyze the impact of both the support structure and the addition of the rotating blades. The experiments were performed in the small-scale testing platform in the hydraulic flume facility at Bucknell University. For each test case, bed form topology was measured after a three-hour time interval using a traversing two-dimensional bed profiler. During the experiment, scour depth measurements at the front face of the cylindrical support structure were taken to estimate the scour rate. Measurements of the bed form were taken across the width of the test section. Results show that the scour hole dimensions increase in the presence of the MHK device. These dimensions also increase with increasing Reynolds number.

Sunday, November 24, 2013 8:00AM - 9:31AM –

Session A14 Experimental Techniques I: PIV Algorithms 302 - Kenneth Christensen, University of Illinois at Urbana-Champaign

8:00AM A14.00001 Direct calculation of the weighting function and depth of correlation in Micro-Particle Image Velocimetry (Micro-PIV) from particle images , MICHAEL HEIN, Saarland University, MPI for Dynamics and Self-Organization, BERNHARD WIENEKE, LaVision GmbH, RALF SEEMANN, Saarland University, MPI for Dynamics and Self-Organization — Micro-PIV has become the most popular tool to measure flow profiles in microfluidics. When measuring in-plane velocities in a three dimensional flow the measured velocity depends on all particles in the images, even on defocused particles, and is given by a weighted average of the true velocity dx(z) with a weighting function W(z). W(z) depends on the optical setup as well as on the particle diameter and gradients of the flow-profile. The width of W(z) determines the height-extension of the plane in which particles can influence the measurement (Depth of Correlation, DOC). Thus the knowledge of the system dependent W(z) is crucial and can be used to reduce the errors introduced by depth-averaging the velocity field. We determine W(z) and thus the DOC using artificial double images for any given flow profile generated from particle images taken with the same optical setup as used for the PIV measurements. Experimental results for objectives with different numerical apertures (NA), different particle sizes and various out-of-plane gradients will be discussed. The resulting weighting function turns out to be quite asymmetric for air-objectives with high NAs, differing significantly in shape and width (DOC) from existing theoretical predictions.

8:13AM A14.00002 Direct Measurement of Rotation and Scaling in Particle Image Velocimetry using the Fourier-Mellin Transform , MATTHEW GIARRA, Virginia Tech, JOHN CHARONKO, Los Alamos National Laboratory, PAVLOS VLACHOS, Purdue University — Traditional particle image velocimetry (PIV) can fail in the presence of spatial velocity gradients because the shearing, stretching, and rotation of particle image patterns can corrupt Cartesian cross correlations. We propose a novel algorithm that measures the rotation and isotropic scaling of individual subregions of PIV images. Our algorithm adopts the Fourier-Mellin (FM) image transformation, which decouples rotation from isotropic scaling and is invariant to translation. Rotation and scaling in the original image manifest as orthogonal translations in the FM-domain, which can then be measured by standard cross correlation. These properties allow for the direct measurement of vorticity (rotation) within a region of interest without relying on the spatial differencing of adjacent velocity vectors. Our algorithm also improves velocity estimates in regions of large rotation (like vortex cores) by applying the inverse rotation and stretching of the particle pattern prior to performing Cartesian correlations that estimate displacements. In this work, we apply our algorithm to synthetic and experimental PIV images and show significant improvement to the vorticity and velocity estimates compared to traditional PIV in regions where rotation is significant.

8:26AM A14.00003 A general approach for time-supersampling of 3D-PIV data by the vortex-in-cell method[1] , FULVIO SCARANO, JAN SCHNEIDERS, RICHARD DWIGHT, TU Delft, AEROSPACE ENGINEERING/AERODYNAMICS TEAM — Advancements of tomographic PIV [1] have led into 3D time-resolved experiments to study the dynamical evolution of 3D turbulent flows [2]. The known bottleneck of Tomo-PIV is the high laser power required to illuminate large volumes in airflows, which becomes critical beyond 10kHz. Time-super-sampling is an approach to reduce the sampling rate, proven for frozen turbulence where the advection model yields a significant increase of temporal resolution [3]. Instead, in separated flows, the advection principle yields unacceptable distortions. The use of Navier-Stokes numerical calculations with the vortex-in-cell (VIC) method is proposed herein. The assumption is made of inviscid incompressible flow [4]. The spatial-resolution of the data is exploited to increase the temporal resolution. The dynamical evolution of the vorticity and velocity field between subsequent snapshots in the 3D domain is numerically evaluated. The verification with fully time resolved data of a circular jet indicates a substantial increase of temporal resolution. Interestingly, data sampled below the Nyquist limit could be reconstructed faithfully, indicating the potential of VIC in alleviating requirements on PIV measurement rate.

[1]Work supported by the European Research Council grant 202887.

8:39AM A14.00004 Divergence-free filtering and pressure determination from 3D velocimetry: applications to flows of industrial and biomedical relevance , DANIELE SCHIAVAZZI, University of California at San Diego, FILIPPO COLETTI, JULIEN BODART, JOHN K. EATON, Stanford University — Methodologies to acquire three-dimensional velocity fields are becoming increasingly available. However unavoidable experimental errors limit the possibility of exploiting the data to extract further information. We recently introduced a noise reduction algorithm which eliminates spurious divergence in incompressible flow measurements, removing about fifty percent of the Gaussian noise. Here we apply the algorithm to the mean velocity field in an inclined jet in crossflow measured by Magnetic Resonance Velocimetry. The de-noised field is used to calculate the mean pressure distribution by integrating the Reynolds-averaged momentum equation. A simple eddy-viscosity model is used for the estimation of the Reynolds stresses. The results are compared with a highly resolved Large Eddy Simulation of the same configuration. It is argued that filtering of the spurious noise can be critical to obtain a correct evaluation of the pressure field. Applications to biomedical flows are also discussed. Results are presented for in vivo cardiac flow measurements as well as in vitro velocimetry in a model of human airways.

8:52AM A14.00005 A Comparison of 3D3C Velocity Measurement Techniques , RODERICK LA FOY, Virginia Tech, PAVLOS VLACHOS, Purdue University — The velocity measurement fidelity of several 3D3C PIV measurement techniques including tomographic PIV, synthetic aperture PIV, plenoptic PIV, defocusing PIV, and 3D PTV are compared in simulations. A physically realistic ray-tracing algorithm is used to generate synthetic images of a standard calibration grid and of illuminated particle fields advected by homogeneous isotropic turbulence. The simulated images for the tomographic, synthetic aperture, and plenoptic PIV cases are then used to create three-dimensional reconstructions upon which cross-correlations are performed to yield the measured velocity field. Particle tracking algorithms are applied to the images for the defocusing PIV and 3D PTV to directly yield the three-dimensional velocity field. In all cases the measured velocity fields are compared to one-another and to the true velocity field using several metrics.

9:05AM A14.00006 Quantitative PIV measurement in narrow channels , DANA EHYAEI, KEN KIGER, University of Maryland — This work focuses on making quantitative velocity measurements with a large depth-of-focus within a thin-gap channel, typical of Hele-Shaw cells. The inherent difficulty in such flows is due to the large velocity gradient across the gap and effects due to particle migration. In the simplest case of no particle migration, the PIV correlation peak is broadened due to the parabolic velocity profile, with an expected peak value at the maximum centerline velocity. However, there is an inevitable under-estimation that is typically up to 33% of the centerline velocity for all but the smallest particle images and largest displacements, due to particle image size effects. In addition, inertial particle migration within the channel results in a second correlation peak as the particles rapidly move away from the wall. In later times, as the particles reach their equilibrium position, the particles sample only a single velocity value, and present conditions similar to traditional PIV interrogation. A practical procedure is proposed to make PIV quantitative by manipulating the particles to their equilibrium position prior to performing measurements and a reliable PIV measurement under appropriate working conditions is discussed for diffusive Rayleigh-Bénard convection in a Hele Shaw cell.

9:18AM A14.00007 Quantifying large-scale flow structures in the wake of a 2.5 MW wind turbine using natural snowfall[1] , JIARONG HONG, MOSTAFA TOLOUI, SEAN RILEY, MICHELE GUALA, KEVIN HOWARD, LEONARDO CHAMORRO, JAMES TUCKER, FOTIS SOTIROPOULOS, University of Minnesota — The atmospheric inflow conditions around utility-scale turbines and multi-turbine arrayed wind farms remain poorly known, despite ongoing research, resulting in considerable wind plant power loss and increased annual operating costs. Gaining detailed full-scale flow information is constrained by low resolution spatial characterization of the flow field around turbines due to a lack of utility-scale research facilities and a number of technical challenges associated with obtaining measurements. Taking advantage of natural snowfall, we now achieve velocity field measurements in the wake of a 2.5 MW wind turbine at a scale of 36x36 m^2. The spatial and temporal resolutions of the measurements are sufficiently high to quantify the evolution of blade-generated coherent motions, such as the tip and trailing sheet vortices, identify their instability mechanisms, and correlate them with turbine operations, control, and performance. This technique has been further validated by comparing the obtained mean velocity and Reynolds stress profiles, up to 60 m above the ground with sonic anemometer measurements at specific elevations, where less than a 3% and 10% difference were observed, respectively.

[1]Acknowledgement to Department of Energy

Sunday, November 24, 2013 8:00AM - 9:57AM –

Session A16 Biofluids: Physiological I - Computational Studies in Cardiovascular Flows 304 - Martina Bukac, University of Pittsburgh

8:00AM A16.00001 Effect of Trabeculations on the Hemodynamics of Left Ventricle: A Computational Study[1] , VIJAY VEDULA, JUNG-HEE SEO, Johns Hopkins University, RICHARD GEORGE, ALBERT LARDO, Johns Hopkins Medical Institutions, RAJAT MITTAL, Johns Hopkins University — The endocardium of the human left ventricle is not smooth. There are surface trabeculations as well as papillary muscles that protrude deep into the ventricular cavity. However, most models of ventricular hemodynamics ignore the presence of these surface structures and assume a smooth endocardial surface. Several key questions arise regarding the impact of these structures on ventricular hemodynamics. These surface "roughness elements" could enhance mixing and dissipation. Moreover, the interstitial regions within the trabeculae might be prone to flow stasis, and this has implications for ventricular thrombogenesis. In the present study, we use flow simulation to study this issue for CT derived models of normal human left ventricle. We focus on the near-wall dynamics of the flow and employ a number of different diagnostics to examine the flow dynamics and "washout" in this region.

[1]This research is supported by the U.S. National Science Foundation through (NSF) CDI-Type II grant IOS-1124804. Computational resources for some of the simulations were also provided in part through the NSF grant NSF-OCI-108849.

8:13AM A16.00002 Fluid Dynamics of Contrast Dispersion in Coronary Arteries: Mechanism and Implications for Identification of Flow-Limiting Lesions[1] , PARASTOU ESLAMI, JUNG-HEE SEO, ALBERT C. LARDO, RAJAT MITTAL, Johns Hopkins University — Recent coronary computed tomography angiography studies have noted the presence of axial contrast concentration gradients in stenosed coronary arteries, but the mechanism responsible for this phenomenon is not well understood. We use computational fluid dynamics to study intracoronary contrast dispersion and the correlation of concentration gradients with intracoronary blood flow and stenotic severity. Simulations of flow and contrast dispersion in both canonical and patient derived models of the left coronary artery (LCA) are carried out with a prescribed contrast bolus profile, and stenoses of varying severities (0% to 80%) considered. Data from our CFD simulations show the presence of measurable contrast gradients, the magnitude of which is found to decrease monotonically with stenotic severity and increase monotonically with the pressure drop across the stenosis. All simulated cases indicate a strong inverse correlation between contrast gradients and coronary flow rate. The study reveals that contrast gradients are generated by intracoronary advection effects, and therefore, encode coronary flow velocity.

[1]This research is supported by a grant from Coulter Foundation.

8:26AM A16.00003 Hemodynamics and flow-vessel interaction in patient-specific aorta using unified lattice Boltzmann computation and simulation , HUIDAN (WHITNEY) YU, Indiana University-Purdue University, Indianapolis (IUPUI), ZHIQIANG WANG, YE ZHAO, Kent State University, SHAWN D. TEAGUE, Indiana University — Patient-specific blood flow simulation is mainly relying on the utilization of commercial software. Geometrical simplification and approximation are usually made thus weaken the capability to aid clinical diagnose and assessment. We develop a unified computing platform to simulate patient-specific hemodynamics and flow-vessel interaction using lattice Boltzmann method (LBM), which tightly integrates anatomical-structure extraction from imaging data and numerical simulation in one computation mesh structure, where the LBM solves level set equation for image segmentation and Navier-Stokes equation for fluid dynamics respectively. The patient-specific vessel geometry, volumetric ratio of solid versus fluid, and the orientation of the boundary obtained with high accuracy seamlessly feed to the numerical simulation needs. In order to better treat the complex geometry, we specifically develop volumetric lattice Boltzmann scheme which strictly satisfies mass conservation when boundary moves. Validation study is on hemodynamics and flow-vessel interaction in healthy and diseased aortas. Flow rate and structure, pressure and vorticity distribution, as well as wall normal and shear stresses, are revealed in both cases.

8:39AM A16.00004 Coupled Hemodynamic-Biochemical Modeling of Thrombus Formation in Infarcted Left Ventricles[1] , JUNG HEE SEO, VIJAY VEDULA, RICHARD GEORGE, RAJAT MITTAL, Johns Hopkins University — Patients with heart failure (HF) and left ventricular (LV) systolic dysfunction have higher rates of thromboembolic events including embolic stroke and peripheral arterial thrombi. A common cause of arterial emboli in HF patients is myocardial infarction (MI) and subsequent left ventricular thrombus (LVT) formation. Stagnation of blood and endocardial injury are hypothesized to promote the development of LVT. The identification of high risk patients and the pharmacologic prevention of LVT formation are the keys to preventing embolic events. Stratification of patients at risk for LVT formation is currently limited, and primarily based on global assessment of ventricular function and image based assessment of ventricular wall motion. In this study, we explore a method to predict LVT risk using a multi-physics computational model. The blood flow in the left ventricle is simulated by solving the incompressible Navier-Stokes equation using an immersed boundary method and this is coupled to a convection-diffusion-reaction equation based model of platelet activation and coagulation. The results are then correlated with the other hemodynamic metrics such as wall shear stress and residence time to develop quantitative metrics for the LVT risk prediction.

[1]Supported by NSF CDI-Type II grant IOS-1124804, Computational resource by XSEDE NSF grant TG-CTS100002.

8:52AM A16.00005 Patient-specific simulation of a trileaflet aortic heart valve in a realistic left ventricle and aorta , ANVAR GILMANOV, TRUNG LE, Saint Anthony Falls Laboratory, University of Minnesota, HENRYK STOLARSKI, Civil Engineering, University of Minnesota, FOTIS SOTIROPOULOS, Saint Anthony Falls Laboratory, Civil Engineering, University of Minnesota — We develop a patient-specific model of the left ventricle consisting of: (1) magnetic-resonance images (MRI) data for wall geometry and kinematics reconstruction of the left ventricle during one cardiac cycle and (2) an elastic trileaflet aortic heart valve implanted in (3) a realistic aorta interacting with blood flow driven by the pulsating left ventricle. Blood flow is simulated via a new fluid-structure interaction (FSI) method, which couples the sharp-interface CURVIB [L. Ge, F. Sotiropoulos, JCP, (2007)] for handling complex moving boundaries with a new, rotation-free finite-element (FE) formulation for simulating large tissue deformations [H. Stolarski, A. Gilmanov, F. Sotiropoulos, IJNME, (2013)] The new FE shell formulation has been extensively tested and validated for a range of relevant problems showing good agreements. Validation of the coupled FSI-FE-CURVIB model is carried out for a thin plate undergoing flow-induced vibrations in the wake of a square cylinder and the computed results are in good agreement with published data. The new approach has been applied to simulate dynamic interaction of a trileaflet aortic heart valve with pulsating blood flow at physiological conditions and realistic artery and left ventricle geometry.

9:05AM A16.00006 Computational study of the effect of dynamic wall confinement on ventricular filling , XUDONG ZHENG, QIAN XUE, University of Maine — Ventricular filling is a major cardiac phase in which the freshly oxygenated blood in the left atrium (LA) enters the left ventricle (LV). There is an increasing consensus that dynamics of transmitral blood flow during filling plays a critical role in dictating overall cardiac health and predicting early changes in cardiac function. The ventricular flow during filling is determined by the interplay of incoming mitral jet and myocardial wall confinement and manifested by a complex morphing pattern of an asymmetric vortex ring. In the current study, we employ computational simulations to explore the effects of dynamic wall confinement on ventricular flow in an idealized left ventricle model. The effects of radial and longitudinal confinement as well as wall motion will be investigated, with special interests on vortex dynamics, such as vortex ring tilting, pinch off and breakdown, intraventricular pressure drop, filling velocity, energy dissipation and blood mixing.

9:18AM A16.00007 Pulsatile flow through idealized trabeculae , NICHOLAS BATTISTA, LAURA MILLER, University of North Carolina at Chapel Hill — Trabeculae begin to form in the human developing heart for Reynolds numbers on the order of 10. Other hearts, such as the squid heart, have trabeculae for Re on the order of 10 and larger. The effect of trabeculae on the flow in this range of Re is not well understood. In this study, computational fluid dynamics is used to quantify the effects of Reynolds number and idealized trabeculae height on the resulting flows. An adaptive and parallelized version of the immersed boundary method (IBAMR) is used to solve the fluid-structure interaction problem. We see the formation of vortices depends upon Re and trabeculae height. We then explore how the periodicity of the flow effects vortex formation and shear patterns. This is important because it is thought that these dynamic processes are important to the generation of shear at the endothelial surface layer and strains at the epithelial layer, which will aid in proper development and functionality.

9:31AM A16.00008 Influence of the heart rate and atrioventricular delays on vortex evolution and blood transport inside the left ventricle[1] , SAHAR HENDABADI, Ilinois Institute of Technology, PABLO MARTINEZ-LEGAZPI, University of California, San Diego, YOLANDA BENITO, JAVIER BERMEJO, Cardiology Department, Hospital Gregorio Maranon, Madrid, Spain, JUAN CARLOS DEL ALAMO, University of California, San Diego, SHAWN SHADDEN, University of California, Berkeley — Cardiac resynchronization therapy (CRT) is used to help restore coordinated pumping of the ventricles by overcoming delays in electrical conduction due to cardiac disease. This is accomplished by a specialized cardiac pacemaker that is able to adjust the atrioventricular (AV) delay.A major clinical challenge is to adjust the pacing strategy to best coordinate the blood flow mechanics of ventricular filling and ejection. To this end, we have studied the difference in the vortex formation and its evolution inside the left ventricle (LV) for 4 different AV delays in a cohort of patients with implanted pacemakers. A reconstruction algorithm was used to obtain 2D velocity over the apical long-axis view of the LV from color Doppler and B-mode ultrasound data. To study blood transport, we have identified Lagrangian coherent structures to determine moving boundaries of the blood volumes injected to the LV in diastole and ejected to the aorta in systole. In all cases, we have analyzed the differences in filling and ejection patterns and the blood transport during the E-wave and A-wave formation.Finally we have assessed the influence of the AV delay on 2 indices of stasis, direct flow and residence time.The findings shed insight to the optimization of AV delays in patients undergoing CRT.

[1]NIH award 5R21HL108268 and grants PIS09/02603 and RD06/0010 from the Plan Nacional de Investigacion Cientifica, Spain

9:44AM A16.00009 Hemodynamic consequences of LPA stenosis in single ventricle stage 2 LPN circulation with automatic registration , DANIELE E. SCHIAVAZZI, ETHAN O. KUNG, University of California at San Diego, ADAM L. DORFMAN, University of Michigan - C.S. Mott Children's Hospital, TAIN-YEN HSIA, Great Ormond Street Hospital for Children, ALESSIA BARETTA, Politecnico di Milano, GREGORY ARBIA, INRIA Paris, ALISON L. MARSDEN, University of California at San Diego — Congenital heart diseases such as hypoplastic left heart syndrome annually affect about 3% of births in the US alone. Surgical palliation of single ventricle patients is performed in stages. Consequently to the stage 2 surgical procedure or other previous conditions, a stenosis of the left pulmonary artery (LPA) is often observed, raising the clinical question of whether or not it should be treated. The severity of stenoses are commonly assessed through geometric inspection or catheter in-vivo pressure measurements with limited quantitative information about patient-specific physiology. The present study uses a multiscale CFD approach to provide an assessment of the severity of LPA stenoses. A lumped parameter 0D model is used to simulate stage 2 circulation, and parameters are automatically identified accounting for uncertainty in the clinical data available for a cohort of patients. The importance of the latter parameters, whether alone or in groups, is also ranked using forward uncertainty propagation methods. Various stenosis levels are applied to the three-dimensional SVC-PA junction model using a dual mesh-morphing approach. Traditional assessments methodologies are compared to the results of our findings and critically discussed.

Sunday, November 24, 2013 8:00AM - 9:57AM –

Session A17 Biofluids: Locomotion I - Swimming and Flapping 305 - Michael Plesniak, George Washington University

8:00AM A17.00001 On the hydrodynamics of fish schooling[1] , IMAN BORAZJANI, MOHSEN DAGHOOGHI, University at Buffalo, SUNY — A Considerable number of fish species swim in a coordinated manner within approximately constant and equal distance from each other, forming a pattern which is referred to as a fish school. It is believed that fish schooling results in more efficient swimming. However, no experimental evidence has conclusively shown the hydrodynamic effects of neighboring fish on swimming, probably due to the challenges involved in measuring the performance under controlled conditions in a school. We investigate possible hydrodynamical effects of fish schooling by constructing an infinite school of virtual swimmers based on a mackerel fish body and carangiform kinematics. We carry out our self-propelled simulation based on prescribed undulations of the fish body (assuming that all of the fish in the school move in exact same manner) and calculating motion of the center of mass. One of the most important geometrical factors of the fish schooling pattern seems to be the distance between two adjacent fish in the school. Therefore, we examined fish schools with different distances of two adjacent fish.

[1]This work was partly supported by the Center for Computational Research (CCR), University at Buffalo.

8:13AM A17.00002 The lateral line system of fish as a "hydrodynamic antenna" , JUN ZHANG, LEIF RISTROPH, Courant Institute, New York University, JAMES LIAO, University of Florida, Whitney Laboratory for Marine Bioscience — The lateral line of fish is a specialized flow detection system comprised of pressure- and shear-responsive sensors distributed over the body surface. Here, we explore how the arrangement of these sensors is related to the hydrodynamic information contained in flows. Using a cast model of a rainbow trout placed in a water tunnel, we devise ways to mimic the flows encountered by swimming fish while measuring the near-body flow field. Comparing our results to anatomical studies indicates that the lateral line sensors are well positioned to detect temporal and spatial changes in flow signals. These findings support a view of the lateral line as a "hydrodynamic antenna" that allows sophisticated behaviors such as rheotaxis and prey detection and tracking.

8:26AM A17.00003 Reduced-order model of fish-like swimming due to shedding of unsteady point vortices , PHANINDRA TALLAPRAGADA, Clemson University — Reduced order models of biomimetic swimming in an ideal fluid, relying on the shedding of point vortices at short intervals of time, are useful to illuminate the essential underlying dynamics of locomotion in fluids. However these reduced order models still possess a state space that is very high dimensional, thus presenting challenges to develop control algorithms. A two-dimensional model that fully couples the motion of the solid boundary and the fluid containing singular distributions of vorticity is presented. The model relies on the shedding of unsteady point vortices, from the tip of a fish-like hydrofoil, in place of many steady point vortices. The subsequent reduction in the dimension of the state space makes the model more amenable to control algorithms. A simple case of the heading-angle control of a fish-like body will be illustrated. The model also has the advantage of being computationally significantly less demanding. More interestingly from a theoretical point of view, the reduced order model illustrates the connection between vortex shedding and velocity constraints encountered in rigid body mechanics.

8:39AM A17.00004 Flow Structures and Efficiency of Swimming Fish school: Numerical Study , YUZURU YATAGAI, Graduate School of Information Sciences, Tohoku University, YUJI HATTORI, Institute of Fluid Science, Tohoku University — The flow structure and energy-saving mechanism in fish school is numerically investigated by using the volume penalization method. We calculate the various patterns of configuration of fishes and investigate the relation between spatial arrangement and the performance of fish. It is found that the down-stream fish gains a hydrodynamic advantage from the upstream wake shed by the upstream fish. The most efficient configuration is that the downstream fish is placed in the wake. It reduces the drag force of the downstream fish in comparison with that in solo swimming.

8:52AM A17.00005 Computational design of flapping kinematics of a flexible finite-span foil[1]

, SEUNGPYO HONG, Department of Mechanical Engineering, Pohang University of Science and Technology, JINMO LEE, DONGHYUN YOU, Department of Mechanical Engineering, Carnegie Mellon University — While many of the effects of chordwise flexibility of a two-dimensional plate or a foil under pitching motions are revealed in recent computational and experimental research, the effects of flexibility of a three-dimensional foil on the manipulation of wing-tip vortices as well as leading-/trailing-edge vortices are rarely understood. The present study aims at identifying flow physics associated with flapping motions of flexible finite-span foils and the effects of the flapping kinematics and flexibility of the foil on the propulsive performance. The propulsive performance and fluid dynamics of wing-tip vortices leading-edge and trailing-edge vortices associated with the thrust generation are investigated in detail by conducting numerical simulations of flow over a flapping foil with different span-to-chord aspect ratios and bending stiffness using a recently developed coupled immersed boundary method and computational structural dynamics.

[1]Supported by the Office of Naval Research Grant N000141110652 and the National Research Foundation of Korea Grant NRF-2012R1A1A2003699

9:05AM A17.00006 Critical Point Analysis of Unsteady Flow Separation from a Pitching Plate[1]

, FAEGHEH HOOMAN, PAUL S. KRUEGER, SMU — Unsteady flow separation is of interest for force and moment generation by flapping airfoils, but it is often difficult to determine how small differences in the motion lead to differences in the flow field and resulting forces. To better understand the flow evolution during unsteady separation in pitching maneuvers, analysis was performed of two numerical data sets for the pitch-up of a two-dimensional flat plate in a free stream flow with Re=1000 provided by Prof. J.D. Eldredge at UCLA. In each data set, flow was characterized by identifying the first order critical points of the velocity field and their eigenvalues to locate the vortical structures and separation and attachment points as well as the relative locations of these features. The evolution of the flow structure was evaluated quantitatively using a tracking algorithm to pair related critical points in sequential frames. The critical points were further analyzed to understand relationships between the flow configuration and the hydrodynamics including the drag coefficient and lift coefficient. Results from the two data sets will be compared to quantitatively assess the differences in the flow structures.

[1]Support from NSF Grant No. 1115139 is gratefully acknowledged.

9:18AM A17.00007 Volumetric PIV Behind a Flapping Wing in an Incoming Vortex Flow

, OSCAR CURET, CYNDEE FINKEL, KARL VON ELLENRIEDER, Florida Atlantic University, DANIEL BISSELL, TSI — The propulsive surfaces of flying and swimming animals interact with vortices shed by their own bodies or other animals, if they are traveling in groups. The interaction of the propulsive surface with these structured vortices might be fundamental for stability and/or decreasing the cost of transport. In this work, we investigate the wake generated by a flapping wing in an incoming vortex flow. We used a NACA0012 wing model with aspect ratio of 2, and a d-profile cylinder to generated the incoming vortices. The model was tested in a water channel at a Reynolds number of approximately 10,000, which is relevant to many biological swimmers and flyers. The flow structure generated by the flapping wing was measured using three-dimensional Particle Image Velocimetry (3-D PIV). A series of experiments were performed for different Strouhal numbers, $St = fL/U$, where f is the flapping frequency, L is the amplitude of oscillation, and U is the incoming flow speed. We present the 3-D flow field of the flapping wing in an incoming vortex flow and compare it with the structure of a flapping wing with an undisturbed incoming flow.

9:31AM A17.00008 Modeling and Navigation of Artificial Helical Swimmers in Channels[1]

, FATMA ZEYNEP TEMEL, ALPEREN ACEMOGLU, SERHAT YESILYURT, Sabanci University — Recent developments in micro/nanotechnology and manufacturing techniques make use of micro robots for biomedical applications realizable. Controlled in-channel navigation of swimming micro robots is necessary for medical applications performed in conduits and vessels in living bodies. Successful design and control of micro swimmers can be achieved with full understanding of hydrodynamic behavior inside channels and their interaction with channel walls and resultant flows. We performed experimental and modeling studies on untethered mm-sized magnetic helical swimmers inside glycerol-filled rectangular channels. In experiments it is observed that rotation of swimmers in the direction of helical axis leads to forward motion due to fluidic propulsion and lateral motion due to traction forces near the wall. Effects of surface roughness, swimming direction and rotation frequency on the swimmers' speed are analyzed. The flow induced by the tail motion is visualized using micro-particle image velocimetry and analyzed at different radial positions using Computational Fluid Dynamics models. Results indicate that at low frequencies traction forces are effective, however as frequency increases fluid forces become dominant and fluid flow is affecting the swimming motion of helical swimmers.

[1]We acknowledge the support from TUBITAK (Techonological & Research Council of Turkey) under the grant no: 111M376.

9:44AM A17.00009 Forward and backward motion of artificial helical swimmers in cylindrical channels[1]

, ALPEREN ACEMOGLU, FATMA ZEYNEP TEMEL, SERHAT YESILYURT, Sabanci University — Motion of micro swimmers in confined geometries such as channels is important due to its relevance in in vivo medical applications such as minimally invasive surgery and drug delivery. Here, swimmers with diameters 0.8 mm and lengths 2 to 3 mm are produced with a 3D printer and cylindrical $Nd_2Fe_{14}B$ magnets are placed inside the bodies. Rotating external magnetic field is used for the actuation of artificial swimmers. Different body and tail geometries are produced and experiments are conducted with a glycerol filled circular channel. Result demonstrate that decreasing channel diameter directly affects the forward motion of the swimmer due to the increasing drag. It is observed that step-out frequency, which defines maximum frequency at which the swimmer can establish a synchronous rotation with the external magnetic field, depends on the geometry of the swimmer and the channel diameter. There are significant differences between low and high frequency motion and forward and backward swimming. Longer tails enable higher forward velocities in high frequencies than backward ones, whereas forward and backward velocities are approximately the same at low frequencies. Furthermore backward motion is more stable than the forward one; at high frequencies, swimmers travel almost at the center of the channel for backward motion, and follow a helical trajectory near the wall during the forward motion. According to simulation results there is a flow which is induced by the rotation of the swimmer rotation that affects the swimmer's trajectory.

[1]We acknowledge the support from TUBITAK (Techonological & Research Council of Turkey) under the grant no: 111M376.

Sunday, November 24, 2013 8:00AM - 9:57AM –

Session A18 Biofluids: General I - Vesicle Modeling and Simulations

306/307 - Simon Mendez, Centre national de la recherche scientifique (CNRS)

8:00AM A18.00001 Three-Dimensional Immersed Interface Method Based Vesicle Simulations

, PRERNA GERA, DAVID SALAC, University at Buffalo SUNY — Solving the Stokes equations for a multi-phase system with an embedded in-extensible interface is crucial for understanding vesicle dynamics. In this talk the Immersed Interface Method is used to solve the Stokes equations across an in-extensible interface. The full jump conditions for a piecewise constant viscosity have been developed and will be presented. An implicit linear system is created to obtain the velocity, pressure and tension fields. Preconditioning strategies needed to ensure convergence of this linear system will be also be presented. Convergence analysis indicates that the accuracy of the method equals the underlying discritization, despite the presence of discontinuous solution fields.

8:13AM A18.00002 Numerical simulations of capsules and red blood cells under flow in complex geometries at non-zero Reynolds numbers , SIMON MENDEZ, ETIENNE GIBAUD, JULIEN SIGUENZA, FRANCK NICOUD, I3M (University of Montpellier II - CNRS), 34095 Montpellier, ACSIOM - I3M TEAM — Numerical simulation of flows of vesicles, capsules and cells is a growing field (Misbah 2012). With the objective of understanding the complex fluid-structure interactions involved in such flows, studying microcirculation and suspension rheology or improving drug vectorization, numerous research groups have developed numerical methods to compute the dynamics of deformable objects like capsules and red blood cells, composed by a drop of liquid enclosed by a membrane. However, the most mature methods rely on boundary integrals, the use of which is allowed by the Stokes flow hypothesis: boundary integral method (BIM) is thus an efficient tool to study microfluidics and microcirculation. In some flows, in particular in some medical devices, the Reynolds number may be high, which precludes the use of the BIM. In this talk, we will show how the immersed boundary method can be implemented in an unstructured finite-volume solver to tackle such flows of deformable objects. The method will be detailed and specific attention will be devoted to the validation of the solver, in particular in 2D, where reference results are scarce. Finally, applications of the method to flows of isolated cells will be shown. Reference: Misbah 2012. J. Phys.: Conf. Series 392 (2012) 012005

8:26AM A18.00003 Phase-Field Modeling of Lipid Vesicles With Pores , SAMAN SEIFI, DAVID SALAC, University at Buffalo SUNY — The formation and annihilation of pores in a lipid vesicle membrane is critical to a number of biotechnologies, such as drug delivery. Previous models of vesicle behavior have ignored the influence of topological changes in the vesicle membrane. Here the entire Helfrich model of a vesicle membrane is considered. Topological changes in the vesicle membrane, such as the formation of a pore, are captured through the use of an embedded phase-field model. The numerical method and sample results will be presented.

8:39AM A18.00004 Asymmetric Instability, Symmetric Instability, and Pearling of a Vesicle in Extensional Flow[1] , ANDREW SPANN, VIVEK NARSIMHAN, ERIC SHAQFEH, Stanford University — A vesicle placed in extensional flow can undergo a transition where the vesicle forms a dumbbell shape connected by a thin long neck. We will examine cases where the vesicle shows either symmetric or asymmetric behavior depending on the flow conditions. We present 3D boundary integral simulations for vesicles in planar and uniaxial extensional flows. For high reduced volumes (at least 0.745 for matched inner/outer viscosity vesicles), a stable steady state shape exists for the vesicle at extensional flows of any capillary number, and furthermore this steady state shape approaches an ellipse as capillary number is increased. For lower reduced volume vesicles the equilibrium shape becomes nonconvex and there exists a critical capillary number above which odd perturbations to the vesicle shape drive an asymmetric elongation transition. For vesicles with reduced volume below 0.6, a symmetric elongation transition exists where the neck thins continuously and the vesicle has no steady shape above a critical capillary number. At sufficiently high capillary number we can see the formation of pearls along the neck of the elongating vesicle. We demonstrate that the rate at which flow is increased can affect the number and position of pearls in this phenomenon.

[1]This work was supported by the US Army High Performance Computation Research Center (AHPCRC) and the National Science Foundation (Grant No. CBET-1066263).

8:52AM A18.00005 Lateral migration of a 3D elastic capsule in a Poiseuille flow , BOYOUNG KIM, HYUNG JIN SUNG, Korea Advanced Institute of Science and Technology, KOREA ADVANCED INSTITUTE OF SCIENCE AND TECHNOLOGY TEAM — The lateral migration of a 3D elastic capsule undergoing large deformation in a 3D Poiseuille flow was explored at moderate Reynolds number ($10 \leq Re \leq 100$) as a function of the initial lateral position (y_0), Reynolds number (Re), aspect ratio (ε), viscosity ratio (λ), membrane stretching coefficient (φ) and bending coefficient (γ). Several numerical methods were used to simulate the problem: the immersed boundary method for fluid-structure interaction, the penalty method for volume conservation in the capsule and the front-tracking method for distinguishing the fluid in capsule from the fluid outside capsule. Three different types of capsule motions were observed: tank-treading (TT) motion, tumbling (TU) motion and swinging (SW) motion according to variations of ε and Re. The initial behavior of the elastic capsule was influenced by the initial lateral position (y_0), but the equilibrium position and the dynamic motion of the capsule were not affected by such variations. The capsule had a strong tendency toward TU motion at higher values of Re, φ and γ, whereas the capsule underwent TT or SW motion as the values of ε and λ increased.

9:05AM A18.00006 The Electrohydrodynamics of Lipid Bilayer Vesicles in AC and DC Fields , LANE MCCONNELL, University of New Mexico, PETIA VLAHOVSKA, Brown University, MICHAEL MIKSIS, Northwestern University — Vesicles, which are closed, fluid-filled lipid bilayers, provide an ideal model to study cellular electro and hydrodynamics. Recent experiments and small deformation analysis of vesicles exposed to an electric field have revealed several interesting phenomena, including transitions from oblate to prolate ellipsoidal shapes and poration of the vesicle membrane. Here we use the boundary integral method to numerically investigate the dynamic behavior of a vesicle in various electric field types, including a DC field, an AC field, and a combination of the two. The vesicle membrane is modeled as an infinitely thin, capacitive, area-incompressible interface, with the surrounding fluids presumed to act as leaky dielectrics which allow for charge advection. Vesicle dynamics are determined by balancing the viscous, elastic, and electric stresses on the membrane. We present a comparison of the full nonlinear numerical results with small deformation theory and recent experimental data, then analyze our results in the relevant parameter space and discuss the role of symmetry in the problem.

9:18AM A18.00007 Equilibrium electrodeformation of a vesicle in an ac electric field , YUAN-NAN YOUNG, HERVE NGANGUIA, New Jersey Institute of Technology — Under an ac electric field the equilibrium shape of a vesicle (closed liposome) depends on various physical parameters, such as the electric field frequency, mismatch in fluid conductivities and permittivities. In this work we use a spheroidal model to investigate these dependences. We derive the transmembrane potential for a leaky dielectric spheroidal shell and compute the equilibrium spheroidal shape. When compared with experiments and previous small-deformation analysis, we found that the spheroidal model agrees better with the experiments. In particular the spheroidal model allows for asymptotic analysis on the cross-over frequency between prolate and oblate vesicles, and the comparison with experiments near the cross-over shows that the spheroidal model captures the prolate-oblate transition better than the small-deformation theory.

9:31AM A18.00008 Electrohydrodynamics of Three-Dimensional Vesicles , EBRAHIM KOLAHDOUZ, DAVID SALAC, University at Buffalo SUNY — A new numerical method is presented to model the dynamic behavior of three-dimensional vesicles in the Stokes regime and in the presence of electric fields. The interface is described using the Jet Level Set method of Nave et. al, while a multi-step projection method is used to simultaneously enforce fluid and interface conditions. The electric field is obtained through a second-order Immersed Interface Method, for which the necessary jump conditions have been developed. The fluid equations are solved for using a Continuum Surface Force method. The formulation and a parallel implementation will be presented, in addition to sample results.

9:44AM A18.00009 Deformation of biomimetic membranes under electroporation using DC electric pulses , PAUL SALIPANTE[1], PETIA VLAHOVSKA, Brown University — Electrohydrodynamics of vesicles (closed bilayer membranes) made of lipids or polymers are investigated under strong DC pulses. When a uniform electric field is applied across a membrane, free charges accumulate on both sides of the membrane and the membrane acts as a capacitor. While the membrane is charging, the vesicle deforms into either an oblate or prolate ellipsoid depending on the bulk fluids conductivities. However, once the membrane is fully charged the vesicle adopts a prolate shape. In strong DC pulses, typically used in cell electroporation, the electric stress can induce pores in both lipid and polymer membranes. The instability short-circuits the membrane capacitor, leading to non-ellipsoidal shape and vesicle collapse. The evolution of vesicle shape and the effect of poration is experimentally studied for DC pulses of different strength and duration. Vesicle shape is related to the critical threshold for membrane poration. Membrane composition is varied to observe the effect of membrane viscosity, membrane capacitance, and poration threshold. The transient response of the vesicle, in particular vesicle collapse, is shown to be sensitive to membrane viscosity.

[1]Current Affiliation: National Institute of Standards and Technology

Sunday, November 24, 2013 8:00AM - 9:18AM –

Session A19 Turbulence Modeling I 310/311 - Naseem Ansari, University of Pittsburgh

8:00AM A19.00001 A framework for Large Eddy Simulation (LES) based on spatiotemporal statistical information , PRAKASH VEDULA, PETER ATTAR, ALLEN LABRYER, University of Oklahoma — We present a computational framework that will have the potential to not only improve the efficiency of computational predictions based on LES but will also be able to address a major drawback of many existing constructs of LES, namely inaccurate predictions of the underlying spatiotemporal structure. The latter drawback could be especially critical in prediction of tornado paths and jet-noise intensities. In our proposed framework, the relevant sub-grid scale stress models are constructed based on information that is consistent with the underlying spatiotemporal statistics. Unlike in many existing constructs of LES, the proposed sub-grid scale stress models include non-Markovian or memory terms whose origins can be explained based on the theory of optimal prediction. These optimal models for LES are studied using a one-dimensional Burgers equation with and without forcing. Results indicate that the proposed framework performs better than most existing frameworks of LES, by virtue of accurate predictions of spatiotemporal structure. The presence of coarse-grained temporal information in our sub-grid scale models also allows for faster simulations by allowing for larger time steps. Implications of these findings to more complicated turbulent flows will also be discussed.

8:13AM A19.00002 A comparison of dynamic procedures for subgrid stresses in low- and high-speed channels[1] , SUNGMIN RYU, GIANLUCA IACCARINO, Stanford University — We present a novel dynamic procedure to determine the space- and time-dependent model coefficient of a subgrid scale (SGS) eddy-viscosity model. As a preceding step, an exact relationship between SGS Reynolds stresses and four different types of SGS closures is derived from the incompressible Navier-Stokes equation applying Reynolds decomposition and its filtered relation. To validate the proposed dynamic procedure, large eddy simulations (LES) of freely decaying isotropic turbulence and incompressible turbulent channel flow at $Re_\tau = 395$ are performed with the Vreman model with the model coefficient dynamically determined by the SGS Reynolds stress based procedure. Moreover, LES with the dynamic Smagorinsky model and the Vreman model with the constant coefficient are also carried out to assess the performance of our dynamic model. Finally, LES with our dynamic model in compressible turbulent channel flows is evaluated to demonstrate that the relation of identity derived from the incompressible flow governing equations is also valid in compressible flows.

[1]This work is funded under NASA Cooperative Agreement NNX11AI41A (Technical Monitor Dr. Stephen Woodruff)

8:26AM A19.00003 Effects of filtering parameter value on simulation results , WEIYUN LIU, J.M. MCDONOUGH, University of Kentucky — Aliasing is a fundamental issue in discrete solutions of the Navier–Stokes equations. It arises from under resolution of numerical approximations as occurs in large-eddy simulation and must be treated with a filter. Two approaches to filtering have been distinguished in the LES context: implicit and explicit. Implicit filtering is formally applied to governing equations without specification of a particular filter, and explicit filtering is performed on computed solutions via a prescribed filter, as in signal processing. With explicit filtering, since filtered velocities are used in subsequent time steps, the aliasing phenomenon can potentially be removed completely; we will focus on this form in the present work. Numerical filters, however, are constructed so as to allow control of the degree of aliasing via parameter values set by the user. We will demonstrate that poor choices of such parameters can result in completely non-physical, yet numerically stable, computed solutions for two widely-used filters, Padé and Shuman, for a problem having abundant experimental data for comparisons.

8:39AM A19.00004 An improved dynamic non-equilibrium wall-model for large eddy simulation[1] , GEORGE ILHWAN PARK, PARVIZ MOIN, Center for Turbulence Research, Stanford University — A non-equilibrium wall-model based on unsteady 3D Reynolds-averaged Navier-Stokes (RANS) equations has been implemented in an unstructured mesh environment. The method is similar to that of the wall-model described by Wang and Moin [Phys. Fluids **14**, 2043–2051, (2002)], but is supplemented by a new dynamic eddy viscosity/conductivity model that corrects the effect of the resolved Reynolds stress (resolved turbulent heat flux) on the skin friction (wall heat flux). This correction is crucial for accurate prediction of the skin friction and wall heat flux. Unlike earlier models, this eddy viscosity/conductivity model does not have a stress-matching procedure or a tunable free parameter, and it shows consistent performance over a wide range of Reynolds numbers. The wall-model is validated against canonical (attached) transitional and fully turbulent flows at moderate to very high Reynolds number: a turbulent channel flow at $Re_\tau = 2000$, an H-type transitional boundary layer up to $Re_\theta = 3300$, and a high Reynolds number boundary layer at $Re_\theta = 31000$. An application to the flow over NACA4412 airfoil is ongoing and hopefully will be presented.

[1]This work was supported by the Winston and Fu-Mei Stanford Graduate Fellowship, NASA Aeronautics Scholarship Program, and NASA under the Subsonic Fixed-Wing Program and the Boeing Company.

8:52AM A19.00005 A dynamic two-level large-eddy simulation method for high Reynolds number flows , REETESH RANJAN, SURESH MENON, Georgia Institute of Technology — We present a dynamic hybrid two-level large-eddy simulation method for high Reynolds number flows. The method combines the two level simulation model with a conventional large eddy simulation model through an additive scale separation operator in a dynamic manner. The two level simulation model performs the scale separation through a large-scale function instead of a spatial filtering used by a conventional large eddy simulation model and therefore, it does not from some of the limitations associated with the spatial filtering. The hybrid method ensures that the two level simulation model provides a dominant contribution in flow regions having sharp gradients and in other regions large-eddy simulation model assumes a dominant role leading to an overall efficient computational method for practical applications. The hybridization is achieved through a dynamic and a spatially smooth blending function based on a characteristic length scale. The dynamic evaluation of the blending function is essential for complex flows where a prior contribution of two-level and large-eddy simulation models can not be estimated accurately. Application of dynamic blending function will be demonstrated by simulating high Reynolds number separating/reattaching flow over a bump in a channel.

9:05AM A19.00006 A new Hybrid Filtered Favre average for compressible LES , MASSIMO GERMANO, Duke University, ANTONELLA ABBÀ, Politecnico di Milano — The statistical study of compressible or variable density turbulent flows is usually performed in terms of the Favre average weighted with the *statistical* density. The usual extension of this approach to the Large Eddy Simulation of compressible turbulent flows is in terms of the Filtered Favre average weighted with the *filtered* density. By consequence the Favre averages are not directly recovered by the statistical average of the Filtered Favre quantities. All that is inconvenient as regards the comparison between RANS and LES and as regards the development of some modeling approaches such as the dynamic modeling procedure and the hybrid RANS/LES strategies. In order to simplify the formulation of compressible LES and to relate more directly the filtered quantities to the statistical ones a new Hybrid Filtered Favre average weighted directly with the *statistical* density is proposed and compared with the usual one.

Sunday, November 24, 2013 8:00AM - 9:57AM –

Session A20 Boundary Layers II: Structure 315 - M. Malik, NASA Langley

8:00AM A20.00001 Large- and very-large-scale motions in wall-bounded flows up to $\delta^+ \approx 2000$[1] , JUAN A. SILLERO, JAVIER JIMÉNEZ, U. Politécnica Madrid — Large- (LSM) and very-large-scale motions (VLSM) are investigated using DNSes of turbulent boundary layers and channels in very long computational domains, reaching $\delta^+ \approx 2000$. Two-dimensional connected regions deviating above or below the local mean by more than a given threshold, are constructed in wall-parallel planes for the velocity components and for the pressure, and probability density functions are computed for their streamwise and spanwise lengths. Exponential tails are observed in the buffer, logarithmic and outer regions, that are characteristic of Poisson accretion processes, suggesting that the structures grow by merging smaller ones. In addition, analysis of the tails provides evidence for the presence of VLSMs in u, whereas only LSMs are present in the transversal velocities (v, w) and in the pressure. Although the size of the structures depends on the thresholding value – chosen here as the fraction of the standard deviation at each height that maximizes the number of regions – it is found that boundary layers are shorter and narrower than channels, in agreement with two-point correlations measurements.

[1]Funded by CICYT, INCITE and ERC.

8:13AM A20.00002 An investigation of the very large scale motions in turbulent pipe flow[1] , LEO HELLSTRÖM, Princeton University, ALEXANDER SMITS, Princeton University, Monash University — The very large sale motions (VLSM) in fully developed pipe flow were characterized using three component, time-resolved Stereoscopic Particle Image Velocimetry (SPIV). The work was conducted at Reynolds number of $50,000$ and $100,000$. In accordance with previous work, the VLSM have a characteristic $10-20R$ in pipe flow, meandering structures, contain $40-60\%$ of the TKE and $30-50\%$ of the Reynolds shear stress. Classical and snapshot POD were performed on the 3C fluctuating velocity field. It is shown that Proper Orthogonal Decomposition (POD) can be used as a low energy filter to extract the VLSM. The POD eigenfunctions and the reconstructed velocity field is then used to investigate the behavior and three-dimensional structure of the VLSM.

[1]Supported under ONR Grant N00014-09-1-0263 and NSF Grant CBET-1064257

8:26AM A20.00003 Hairpin Vortices: Autogeneration and Interaction[1] , DANIEL SABATINO, RIJAN MAHARJAN, ANDREW SANDERS[2], Lafayette College — The regeneration of hairpin vortices is examined in a free-surface water channel where vortices are artificially generated by means of injection in a laminar boundary layer. The process is visualized with dye and hydrogen bubble-wire techniques. The strength of an isolated hairpin required to begin the autogeneration process is established by means of PIV measurements on the symmetry plane. Because hairpins are in close proximity in a fully-turbulent boundary layer, two hairpins are generated at different streamwise locations and allowed to interact at different stages of development. The relative position, strength and maturity of the interacting hairpins that generate secondary vortices are examined. The morphology of the generation process and of the resulting secondary hairpin for both the isolated and interacting cases are discussed and compared to previous work.

[1]Supported by the National Science Foundation under Grant CBET-1040236
[2]Now at Georgia Tech

8:39AM A20.00004 A study of synthetic large scales in turbulent boundary layers[1] , SUBRAHMANYAM DUVVURI, MITUL LUHAR, California Institute of Technology, CASEY BARNARD, MARK SHEPLAK, University of Florida, BEVERLEY MCKEON, California Institute of Technology — Synthetic spanwise-constant spatio-temporal disturbances are excited in a turbulent boundary layer through a spatially impulsive patch of dynamic wall-roughness. The downstream flow response is studied through hot wire anemometry, pressure measurements at the wall and direct measurements of wall-shear-stress made using a novel micro-machined capacitive floating element sensor. These measurements are phase-locked to the input perturbation to recover the synthetic large-scale motion and characterize its structure and wall signature. The phase relationship between the synthetic large scale and small scale activity provides further insights into the apparent amplitude modulation effect between them, and the dynamics of wall-bounded turbulent flows in general. Results from these experiments will be discussed in the context of the critical-layer behavior revealed by the resolvent analysis of McKeon & Sharma (*J Fluid Mech*, 2010), and compared with similar earlier work by Jacobi & McKeon (*J Fluid Mech*, 2011). Model predictions are shown to be in broad agreement with experiments.

[1]The support of AFOSR grant #FA 9550-12-1-0469, Resnick Institute Graduate Research Fellowship (S.D.) and Sandia Graduate Fellowship (C.B.) are gratefully acknowledged.

8:52AM A20.00005 Spatial properties of large-scale structure in a turbulent boundary layer[1] , JINYUL HWANG, JIN LEE, KAIST, SEO YOON JUNG, KAERI, TAMER A. ZAKI, Imperial College London, HYUNG JIN SUNG, KAIST — Direct numerical simulation (DNS) database of a zero pressure-gradient turbulent boundary layer was scrutinized to investigate the spatial distribution of vortices around a streamwise-elongated low-speed structure. The turbulent flow field has been numerically produced by preceding the bypass transition simulation using the isotropic free-stream turbulence, in which the Reynolds number reaches up to $Re_\theta = 3280$. In the present study, the low-speed structures and vortices have been identified and tracked in order to obtain spatial properties of the large-scale structures. The information about the inclination angle, distance and population of individual vortices on the long streamwise structure has been investigated. Finally, the present study provides statistical evidence on the formation of large-scale packet-like structure and its variation along the downstream.

[1]This work was supported by the Creative Research Initiatives (No. 2013-003364) program of the National Research Foundation of Korea (MSIP) and partially supported by KISTI under the Strategic Supercomputing Support Program.

9:05AM A20.00006 ABSTRACT WITHDRAWN –

9:18AM A20.00007 Temporal evolution of Townsend's attached eddies[1] , ADRIAN LOZANO-DURAN, JAVIER JIMENEZ, Universidad Politecnica de Madrid — The temporal evolution of the eddies responsible for the momentum transfer in a turbulent channel are studied using time-resolved DNS data at $Re_\tau = 4000$. The eddies are identified as connected regions of intense tangential Reynolds stress, and tracked in time. Once their evolutions are properly organized, they provide the necessary information to characterize eddies from birth to death. Eddies are born at all distances from the wall, although with higher probability near it, where the shear is strongest. Most of them stay small and do not last for long times. However, there is a family of eddies that become large enough to get attached to the wall while they reach into the logarithmic layer. They can be considered the best candidates for Townsend's attached eddies found until now. They are geometrically self-similar, with sizes and lifetimes proportional to their distance from the wall. Eddies associated with ejections move away from the wall with $dy/dt = u_\tau$, and their base attaches very fast at the beginning of their lives. Conversely, sweeps move towards the wall at $-u_\tau$, and attach later. In both cases, they remain attached for $2/3$ of their lives. In the streamwise direction, eddies are advected and sheared by the local mean velocity.

[1]Funded by ERC, CICYT and Spanish Ministry of Science

9:31AM A20.00008 Homage to Bob Brodkey at 85: ejections, sweeps and the genesis and extensions of quadrant analysis , JAMES WALLACE, University of Maryland — Almost 50 years ago Bob Brodkey and his student, Corino, conceived and carried out a visualization experiment for the very near wall region of a turbulent pipe flow (JFM 37) that, together with the turbulent boundary layer visualization of Kline et al. (JFM 30), excited the turbulence community. Using a high speed movie camera mounted on a lathe bed that recorded magnified images in a moving frame of reference, they observed the motions of small particles in the sub- and buffer-layers. Surprisingly, these motion were not nearly so locally random as was the general view of turbulence at the time. Rather, connected regions of the near wall flow decelerated and then erupted away from the wall in what they called "ejections." These decelerated motions were followed by larger scale connected motions toward the wall from above that they called "sweeps." Brodkey and Corino estimated that ejections accounted for 70% the Reynolds shear stress at $Re_d = 20,000$ while only occurring about 18% of the time. Wallace et al. (JFM 54) attempted to quantify these visual observations by conceiving of and carrying out a quadrant analyisis in a turbulent oil channel flow. This paper will trace this history and describe the expanding use of these ideas in turbulence research today.

9:44AM A20.00009 Probability density function of pressure in turbulent boundary layers , YOSHIYUKI TSUJI, Nagoya University, Japan, YOSHINOBU YAMAMOTO, Yamanashi University, Japan — We study the probability density function (PDF) inside turbulent boundary layer and discuss how their shapes vary depending on the Reynolds number and the distance from the wall. Pressure fluctuations are measured by small pressure probes developed by our research group so far. Experimental data are measured in high Reynolds number facilities. We measured both static pressure and wall pressure simultaneously in turbulent boundary layers up to Reynolds numbers based on the momentum thickness 44000. Direct numerical simulation of channel flow data is used for the present research. It is found that PDF shapes have slightly Re-number dependence but they are self-similar if normalized by their standard deviation. There is no great difference of PDF between channel and boundary layer flows expect near wall region.

Sunday, November 24, 2013 8:00AM - 9:57AM –

Session A21 Turbulence: Simulations I - LES Application 316 - Johan Larsson, University of Maryland

8:00AM A21.00001 Wall-modeled large eddy simulation of high-lift devices from low to post-stall angle of attacks[1] , JULIEN BODART, Universite de Toulouse, ISAE, JOHAN LARSSON, Department of Mechanical Engineering, University of Maryland, PARVIZ MOIN, Center for Turbulence Research, Stanford University — The flow around a McDonnell-Douglas 30P/30N multi-element airfoil at the flight Reynolds number of 9 million (based on chord) is computed using LES with an equilibrium wall-model with special treatment for transitional flows. Several different angles of attack are considered, up to and including stall, challenging the wall-model in several flow regimes. The maximum lift coefficient, which is generally difficult to predict with RANS approaches, is accurately predicted, as compared to experiments performed in the NASA LPT wind-tunnel.

[1]NASA grant: NNX11AI60A

8:13AM A21.00002 Wavelet-based adaptive LES of turbulent flow around a square-cylinder , GIULIANO DE STEFANO, University of Naples (Italy), OLEG V. VASILYEV, University of Colorado Boulder — The incompressible turbulent flow around a two-dimensional bluff body with square cross-section is simulated by using a wavelet-based adaptive LES method. The presence of the obstacle is modeled with the Brinkman volume-penalization technique, which results in modifying the governing equations with the addition of an appropriate forcing term inside the spatial region occupied by the cylinder. The localized dynamic kinetic-energy-based approach (De Stefano et al., PF 2008) is utilized to model the residual stresses term in the wavelet-filtered volume-penalized incompressible Navier-Stokes equations. The filtered momentum and SGS energy equations are numerically solved by means of the adaptive wavelet collocation method, where the time-dependent non-uniform spatial grid is dynamically determined following the flow evolution. The combined volume-penalization/wavelet-collocation approach is successfully applied to the simulation of turbulent vortex shedding flow behind a stationary prism with square cross-section at moderate Reynolds number. The present results are in good agreement with both experimental findings and data from non-adaptive numerical solutions.

8:26AM A21.00003 Numerical Study of Impulse Actuated Stall Control , SIGFRIED HAERING, ROBERT MOSER, University of Texas at Austin — Experimental studies have shown that pulse actuated dynamic stall control provides a simple means to significantly increase the performance of lifting surfaces and expand their flight envelope. However, precise information of the complex boundary layer reattachment mechanisms are inaccessible to experimental measurements. Therefore, adequately resolved and validated simulations are necessary to fully understand and utilize this approach. Numerical studies using detached large eddy simulation (DES) of a stalled airfoil with both spanwise-uniform and staggered actuation regions experiencing single pulse actuated flow reattachment are performed. The results of these simulations provide fundamental insight into the stall control mechanisms observed in experiments. Such studies may be extended to design optimal actuator spacing, orientation, and scheduling.

8:39AM A21.00004 Dynamic Immersed Boundary Method for Modeling of Turbulent Boundary Layers over Bio-Fouled Surfaces[1] , XIANG YANG, JASIM SADIQUE, RAJAT MITTAL, CHARLES MENEVEAU, Johns Hopkins University — The growth of large organisms on ship surfaces, i.e. macrobiofouling, is a major contributor to drag, and consequently, fuel consumption. The problem of turbulence over biofouled surfaces may be reduced to that of a developing turbulent boundary layer over a surface with a wide range of roughness length scales. Due to the presence of these scales, direct numerical simulation (DNS) or even wall-resolved large-eddy-simulation (LES) is prohibitively expensive. We address this challenge by developing a dynamic immersed boundary method that does not require the flow field nor the roughness to be fully resolved. The effect of unresolved small eddies are included via an LES sub-grid model. The large-scale roughness elements are resolved by a sharp-interface immersed boundary method and the effect of small (unresolved) roughness elements is incorporated through the use of a wall model that assumes a log-law at the grid point closest to the wall. This computationally efficient method is validated against experiments of developing turbulent boundary layer with multiple-scale roughness elements. We present results from this study and provide a discussion of our findings.

[1]This work is funded by the Office of Naval Research (ONR) grant N00014-12-1-0582. Fruitful interactions with M. Schultz (USNA), B. Ganapathisubramani and M. Placidi (Southhampton) are also gratefully acknowledged.

8:52AM A21.00005 Computational Analysis of Particle Nucleation in Dilution Tunnels: Effect of Flow Configuration and Tunnel Geometry , SATBIR SINGH, PETER ADAMS, ASHWIN MISQUITTA, KYUNG LEE, Carnegie Mellon University, ERIC LIPSKY, Penn State Greater Allegheny, ALLEN ROBINSON, Carnegie Mellon University — Measurement of fine particle emission from combustion sources is important to understand their health effects, and to develop emissions regulations. Dilution sampling is the most commonly used technique to measure particle number distribution because it simulates the cooling of combustion exhaust with atmospheric air. Experiments suggest that the measured distribution is dependent on the dilution ratio used and the tunnel design. In the present work, computational analysis is performed to investigate the effect of tunnel flow and geometric parameters on H_2SO_4-H_2O binary nucleation inside dilution tunnels using a large-eddy-simulation (LES) based model. Model predictions suggest that the experimental trends are likely due to differences in the level of turbulence inside the tunnels. It is found that the interaction of dilution air and combustion exhaust in the mixing layer greatly impacts the extent of nucleation. In general, a cross-flow configuration with enhanced turbulent mixing leads to greater number of nucleation-mode particles than an axial-flow configuration.

9:05AM A21.00006 Coherent eddies in flows over three-dimensional dunes , MOHAMMAD OMIDYEGANEH, UGO PIOMELLI, Queen's University — We performed large-eddy simulations of the flow over a series of 3D dunes at laboratory scale. The bedform three-dimensionality was imposed by shifting a standard 2D dune shape in the streamwise direction according to a sine wave. The flow structures are discussed for two cases, with the same crestline amplitudes and wavelengths but different crestline alignments: in-phase and staggered. Large-scale, mean streamwise vortices are the primary factor that alters the features of the instantaneous flow structures. Rollers generated in the separated shear layer appear regularly over the lobe, and are shed more frequently than in 2D geometries. Separated vortices in the lobe plane undergo a three-dimensional instability while advected downstream, and rise toward the free surface while developing into a horseshoe shape (similar to the 2D case). When the tip of such a horseshoe reaches the free surface, the ejection of flow at the surface causes boils (upwelling events on the surface). Strong boil events are observed on the surface of the lobe planes of 3D dunes more frequently than in the saddle planes and the corresponding 2D geometry. Boil events occur at higher frequency in the staggered alignment, but with less intensity than in the in-phase alignment.

9:18AM A21.00007 LES study of vortical structures and suction peaks on a 3D square cylinder in turbulent boundary layer , TETSURO TAMURA, Tokyo Institute of Technology, YOSHIYUKI ONO, Obayashi Corporation — Sophisticated LES technique has made it possible to reproduce unsteady flows around a three-dimensional square cylinder in turbulent boundary layer. Various flow patterns such as the separation bubble or vortices in the cylinder wake are sensitively changed depending on the angle of attack. It is well-known that local severe suctions occur in the flow separation regions of the cylinder. The experimental study showed that the local severe suctions on the side of the square cylinder were caused by two types of conical vortices. One was a standing conical vortex at the upper edge of windward corner at glancing angle, which leads to large level of negative pressure. The other was an inversion conical vortex on the lower side of the cylinder when the flow normally attacks. This inversion conical vortex was periodically formed and causes the fluctuation of the pressure near the bottom of windward corner. Here, LES is applied to the flow around a square cylinder in boundary-layer turbulence. Instantaneous large negative pressure peaks are randomly recognized in the present computational results. At the same time, inversion conical vortex and conical vortices are intermittently formed in boundary-layer turbulence. Physical mechanism for occurrence of peak pressures has been elucidated.

9:31AM A21.00008 Turbulent Heat Transfer in Curved Pipe Flow[1] , CHANGWOO KANG, KYUNG-SOO YANG, Inha University, Korea — In the present investigation, turbulent heat transfer in fully-developed curved pipe flow with axially uniform wall heat flux has been numerically studied. The Reynolds numbers under consideration are $Re_\tau = 210$ (DNS) and 1,000 (LES) based on the mean friction velocity and the pipe radius, and the Prandtl number (Pr) is 0.71. For $Re_\tau = 210$, the pipe curvature (κ) was fixed as 1/18.2, whereas three cases of κ (0.01, 0.05, 0.1) were computed in the case of Re_τ =1,000. The mean velocity, turbulent intensities and heat transfer rates obtained from the present calculations are in good agreement with the previous numerical and experimental results. To elucidate the secondary flow structures due to the pipe curvature, the mean quantities and rms fluctuations of the flow and temperature fields are presented on the pipe cross-sections, and compared with those of the straight pipe flow. To study turbulence structures and their influence on turbulent heat transfer, turbulence statistics including but not limited to skewness and flatness of velocity fluctuations, cross-correlation coefficients, an Octant analysis, and turbulence budgets are presented and discussed. Based on our results, we attempt to clarify the effects of Reynolds number and the pipe curvature on turbulent heat transfer.

[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).

9:44AM A21.00009 Secondary Peak in Nusselt Number for Jet Impinging Flows: LES Study , RABIJIT DUTTA[1], ANUPAM DEWAN[2], BALAJI SRINIVASAN[3], Department of Applied Mechanics, Indian Institute of Technology Delhi, New Delhi-110016 — Jet impingement heat transfer is widely studied because of its industrial and as well as fundamental relevance. A secondary peak in Nusselt number at some distance away from the stagnation point is observed for both round and slot jet impingement flows at a small nozzle-to-plate spacing. Although various researchers have studied the reason behind this secondary peak, it is still an open question in the literature. We present large eddy simulation (LES) of turbulent slot jet impingement heat transfer to gain further insight into the phenomenon of secondary peak in Nusselt number. Profiles of mean velocities, turbulent fluctuating velocities and Nusselt numbers have been analyzed along the impingement plate. A sudden increase in the wall normal turbulence fluctuations have been observed in the region where the secondary peak in Nusselt number occurs. Further, an analysis of vortex structures of the flow showed that the increase in the wall normal turbulence could be associated with the secondary vortex observed near the impingement wall.

[1]PhD candidate
[2]Professor
[3]Assistant Professor

Sunday, November 24, 2013 8:00AM - 9:05AM –

Session A22 Turbulence Modeling II 317 - Ralph Metcalfe, University of Houston

8:00AM A22.00001 By-pass mechanism for transition to turbulence in supercritical pipe flow , RONALD J. ADRIAN, Arizona State University, XIAOHUA WU, Royal Military College of Canada, PARVIZ MOIN, Center for Turbulence Research, Stanford University, JON R. BALTZER[1], Arizona State University, JEAN-PIERRE HICKEY, Center for Turbulence Research, Stanford University — Direct numerical simulations of 250R long pipe flows evolving spatially at Reynolds number of 6000 and 8000 show that a thin ring of turbulent fluctuations extracted from a fully turbulent simulation superimposed on the linearly stable parabolic profile is capable of causing transition to the self-sustaining turbulent state. The finite amplitude disturbances from the ring create a roughly annular turbulent wake that grows downstream radially inward and outward. Transition is triggered by vortex filaments drawn from the disturbance region, intensified by stretching, and moving towards the wall. Between 30-40R the filaments induce inward radial flows; in turn, they create lambda or hairpin vortices that rapidly form into hairpin packets. Small-scale activity explodes when the packets create turbulent spots and overcomes larger-scale fluctuations from the initial disturbances as it grows and interacts to fill the pipe with turbulence that approaches the fully developed state by about 75R. Following hairpin formation the process is similar to transition in boundary layers. Unlike boundary layers, there is no stage of linear instability in the pipe, and 2-D and 3-D finite amplitude instabilities near the wall do not appear to play a role.

[1]Presently at Los Alamos National Laboratory

8:13AM A22.00002 Following analytically stages of transition in Couette flow , MICHAEL KARP, JACOB COHEN, Faculty of Aerospace Engineering, Technion — A possible explanation for transition in Couette flow is the mechanism of transient growth (TG). Accordingly, a small disturbance can achieve a significant non-modal TG and trigger nonlinear mechanisms before its eventual decay owing to viscosity. The linear optimal disturbance achieving the maximal growth consists of a pair of streamwise independent counter-rotating vortices (CVPs) which create spanwise-varying streamwise streaks. These may become unstable with respect to infinitesimal disturbances. It is shown that four decaying normal modes, obtained analytically, are sufficient to follow the linear TG mechanism. A secondary linear stability analysis of the modified base-flow (Couette flow with streaks) is conducted using Floquet theory for the spanwise periodic base-flow. The predictions of the stability analysis are compared with direct numerical simulations using the "Channelflow" code. It is shown analytically that the inclusion of nonlinear interactions between the base-flow and the CVPs is required in order to predict instability. Furthermore, it is demonstrated that the generation of a 'strong' inflectional point is more important than obtaining maximal growth. The minimal number of modes enables us to follow analytically several key stages of the transition process.

8:26AM A22.00003 Early stages of transition in viscosity-stratified channel flow , RAMA GOVINDARAJAN, SHARATH JOSE, Tata Institute of Fundamental Research, Hyderabad, LUCA BRANDT, KTH Mechanics, Stockholm — In parallel shear flows, it is well known that transition to turbulence usually occurs through a subcritical process. In this work we consider a flow through a channel across which there is a linear temperature variation. The temperature gradient leads to a viscosity variation across the channel. A large body of work has been done in the linear regime for this problem, and it has been seen that viscosity stratification can lead to considerable changes in stability and transient growth characteristics. Moreover contradictory effects of introducing a non uniform viscosity in the system have been reported. We conduct a linear stability analysis and direct numerical simulations (DNS) for this system. We show that the optimal initial structures in the viscosity-stratified case, unlike in unstratified flow, do not span the width of the channel, but are focussed near one wall. The nonlinear consequences of the localisation of the structures will be discussed.

8:39AM A22.00004 Transient growth of disturbances in near-wall region of turbulent channel flow , EUIYOUNG KIM, HAECHEON CHOI, Seoul National University, JOHN KIM, UCLA — The transient growth of optimal disturbances has been suggested as a part of self-sustaining process of turbulent structures. It is generally accepted that the self-sustaining process is independent of the outer part of a boundary layer. In this study, we investigate the relationship between the optimally amplified disturbances in the near-wall region and turbulent structures in turbulent channel flows for $Re_\tau = 180$ to 10000. Optimal disturbances in a confined domain ($0 < y^+ < y_c^+$) are considered and disturbances at $y^+ > y_c^+$ are damped out. The most amplified disturbances in a confined domain are streamwise velocity streaks that are induced by streamwise vortices. The wavelength, growth, and growth time of disturbances are almost independent of Re_τ when they are are normalized by viscous wall units. They increase with the domain height y_c^+. Especially, the optimal disturbance in $0 < y^+ < 40$ shows good agreement with the near-wall structures of turbulent channel flow.

8:52AM A22.00005 Identification of spatially-localized flow structures via sparse proper orthogonal decomposition , NEIL DHINGRA, MIHAILO JOVANOVIC, University of Minnesota, PETER SCHMID, Ecole Polytechnique — Proper Orthogonal Decomposition (POD) has become a standard tool for identification of the most energetic flow structures in fluid flows. It relies on the maximization of a quadratic form subject to a quadratic equality constraint, which can be readily accomplished via a singular value decomposition. For spatially homogeneous (or nearly homogeneous) flows, the resulting flow structures are global (or have large support) in the spatial domain of interest. By augmenting the optimization problem with an additional penalty term that promotes sparsity in the physical space, we are able to obtain energetic flow structures that become increasingly localized as our emphasis on sparsity increases. The resulting optimization problem, formulated in terms of an augmented Lagrangian functional, is solved using the Alternating Direction Method of Multipliers followed by a postprocessing step. The sparse POD algorithm is applied to the linearized Navier-Stokes equations for a plane channel flow, and the emergence of spatially localized structures is observed for increasing penalty terms. This test case and the underlying optimization techniques build the foundation for further studies into the relevance and role of localized perturbations on the overall behavior of general shear flows.

Sunday, November 24, 2013 8:00AM - 9:57AM –

Session A23 Turbulence: Theory I - General 318 - Antonio Attili, King Abdullah University of Science and Technology

8:00AM A23.00001 The definition of turbulence and the direction of the turbulence energy cascade , CARL GIBSON, Univ. of Cal. at San Diego — Turbulence is defined as an eddy-like state of fluid motion where the inertial-vortex forces of the eddies are larger than any other forces that tend to damp the eddies out. Because vorticity is produced at the Kolmogorov scale, turbulent kinetic energy always cascades from small scales to large. Irrotational flows that supply kinetic energy to turbulence from large scale motions are by definition non-turbulent. The Taylor-Reynolds-Lumley cascade of kinetic energy from large scales to small is therefore a non-turbulent cascade. The Reynolds turbulence poem must be revised to avoid further confusion. Little whorls on vortex sheets, merge and pair with more of, whorls that grow by vortex forces, Slava Kolmogorov! Turbulent mixing and transport processes in natural fluids depend on fossil turbulence and fossil turbulence waves, which are impossible by the TRL cascade direction. Standard models of cosmology, astronomy, oceanography, and atmospheric transport of heat, mass, momentum and chemical species must be revised. See journalofcosmology.com Volumes 21 and 22 for oceanographic and astro-biological examples.

8:13AM A23.00002 On Lagrangian and Eulerian Acceleration in Rotating and Sheared Homogeneous Turbulence , FRANK JACOBITZ, University of San Diego, KAI SCHNEIDER, Aix-Marseille Universite, WOUTER BOS, Ecole Centrale de Lyon, MARIE FARGE, Ecole Normale Supérieure — The Lagrangian and Eulerian acceleration properties of turbulence are of importance for problems ranging from fundamental theoretical considerations to modeling of dispersion processes. The acceleration statistics of rotating and sheared homogeneous turbulence are studied here using direct numerical simulations. The study focusses in particular on the influence of the Coriolis to shear rate ratio and also on the scale dependence of the statistics. The probability density functions (pdfs) of both Lagrangian and Eulerian acceleration show a strong and similar influence on the rotation ratio. The flatness further quantifies this influence and yields values close to three for strong rotation. For moderate and vanishing rotation, the flatness of the Eulerian acceleration is larger than that of the Lagrangian acceleration, contrary to previous results for isotropic turbulence. A wavelet-based scale-dependent analysis shows that the flatness of both Eulerian and Lagrangian acceleration increases as scale decreases. For strong rotation, the Eulerian acceleration is more intermittent than the Lagrangian acceleration, while the opposite result is obtained for moderate rotation.

8:26AM A23.00003 Scale-Dependent Stress-Strain Rate Alignment and Spectral Transport in 2D Turbulence , YANG LIAO, NICHOLAS T. OUELLETTE, Yale University, YALE UNIVERSITY TEAM — The flux of quantities such as energy or enstrophy between different scales can be expressed as the scalar product of an appropriate scale-dependent stress and a rate of strain. But regardless of their magnitudes, spectral transfer can be suppressed if the stress and strain rate are geometrically misaligned. Working with experimental data obtained from an experimental quasi-two-dimensional weakly turbulent flow, we explore the impact of geometric alignment on the spectral transfer of energy and enstrophy using filter-space techniques. We decompose the scale-dependent stress into three distinct components, and show that they tend to drive spectral transport in different directions. We also show that the net observed directionality of the inverse energy and forward enstrophy cascades are controlled by the scale-dependet geometric alignment of these quantities.

8:39AM A23.00004 Classical Turbulence Scaling and Intermittency in Strongly Stratified Turbulence[1] , STEVE DE BRUYN KOPS, University of Massachusetts Amherst — A -5/3 slope in the velocity and scalar spectra of stratified turbulence has long been taken as a sign that turbulence in this regime may scale as hypothesized by Kolmogorov, Oboukhov, and Corrsin (KOC). It has also been observed, however, that if the flow is in the strongly stratified regime then the buoyancy force is not insignificant and so some of the assumptions that underlie the KOC scaling hypotheses do not hold. The KOC hypotheses imply more than just -5/3 slope in spectra, though. We consider scaling of the second- and third-order velocity structure functions, the second-order scalar structure function, and the third-order mixed velocity-scalar structure functions. In addition, we examine the scaling of the dissipation rate in light of Kolmogorov's hypotheses on internal intermittency. Direct numerical simulations in the strongly stratified regime with buoyancy Reynolds numbers between 13 and 220 are examined, along with isotropic homogeneous turbulence with similar dynamic range. The simulations are resolved on up to $8192 \times 8192 \times 4096$ grid points. For unstratified turbulence, the dynamic range that these large grids enable is sufficient for KOC scaling to be evident, and for the intermittency exponent to be close to its textbook value.

[1]Supported by the US Office of Naval Research with High Performance Computing resources provided by the US DOD High Performance Computing Modernization Program

8:52AM A23.00005 Scaling of Lyapunov Exponents in Homogeneous, Isotropic DNS , NICHOLAS FITZSIMMONS, NICHOLAS MALAYA, ROBERT MOSER, University of Texas Austin — Lyapunov exponents measure the rate of separation of initially infinitesimally close trajectories in a chaotic system. Using the exponents, we are able to probe the chaotic nature of homogeneous isotropic turbulence and study the instabilities of the chaotic field. The exponents are measured by calculating the instantaneous growth rate of a linear disturbance, evolved with the linearized Navier-Stokes equation, at each time step. In this talk, we examine these exponents in the context of homogeneous isotropic turbulence with two goals: 1) to investigate the scaling of the exponents with respect to the parameters of forced homogeneous isotropic turbulence, and 2) to characterize the instabilities that lead to chaos in turbulence. Specifically, we explore the scaling of the Lyapunov exponents with respect to the Reynolds number and with respect to the ratio of the integral length scale and the computational domain size.

9:05AM A23.00006 Using information theory for turbulence prediction: a statistical approach[1] , WALTER GOLDBURG, RORY CERBUS, University of Pittsburgh — Information theory provides a tool for quantifying the amount of uncertainty or disorder in physical systems through the entropy density h. Going beyond this, physics is often concerned with prediction. The goal here is to predict a subsequent string of velocity measurements on the basis of a set of prior observations. The predictability is captured in a function called the system's statistical complexity C, which is the average information needed for the prediction. There have been very few attempts to use this theory with experimental data. We have measured C in a quasi-2D soap film flow as a function of Reynolds number Re. The measurements point to a sharp transition in C(Re) when the turbulence becomes fully developed. This approach to complexity through predictability promises to be an interesting way of looking at turbulence and other complex systems.

[1]NSF Grant No. 1044105, Okinawa Institute of Science and Technology (OIST)

9:18AM A23.00007 Turbulent scaling laws as solutions of the multi-point correlation equation using statistical symmetries , MARTIN OBERLACK, ANDREAS ROSTECK, VICTOR AVSARKISOV, TU Darmstadt, Dept. Mechanical Engineering — Text-book knowledge proclaims that Lie symmetries such as Galilean transformation lie at the heart of fluid dynamics. These important properties also carry over to the statistical description of turbulence, i.e. to the Reynolds stress transport equations and its generalization, the multi-point correlation equations (MPCE). Interesting enough, the MPCE admit a much larger set of symmetries, in fact infinite dimensional, subsequently named statistical symmetries. Most important, theses new symmetries have important consequences for our understanding of turbulent scaling laws. The symmetries form the essential foundation to construct exact solutions to the infinite set of MPCE, which in turn are identified as classical and new turbulent scaling laws. Examples on various classical and new shear flow scaling laws including higher order moments will be presented. Even new scaling have been forecasted from these symmetries and in turn validated by DNS. Turbulence modellers have implicitly recognized at least one of the statistical symmetries as this is the basis for the usual log-law which has been employed for calibrating essentially all engineering turbulence models. An obvious conclusion is to generally make turbulence models consistent with the new statistical symmetries.

9:31AM A23.00008 Comparing nearly singular vorticity moments in Euler and Navier-Stokes numerical solutions , ROBERT M. KERR, University of Warwick — The inviscid growth of a range of vorticity moments in Navier-Stokes and Euler calculations are compared for simulations using a new anti-parallel initial condition. One goal is to understand the origins of a new hierarchy of rescaled vorticity moments in several Navier-Stokes calculations where the rescaled moments obey $D_m \geq D_{m+1}$, the reverse of the usual $\Omega_{m+1} \geq \Omega_m$ Hölder ordering. Two temporal phases have been identified for the Euler calculations. In the first phase the $1 < m < \infty$ vorticity moments are ordered as $D_m \geq D_{m+1}$, as in the Navier-Stokes case and grow in a manner that skirts possible singular growth with $D_m^2 \rightarrow \sup|\omega| \sim A_m(T_c - t)^{-1}$ with the A_m are nearly independent of m. In the second phase, the new D_m ordering breaks down and the Ω_m converge towards super-exponential growth for all m, shown by plotting $\log(d\log\Omega_m/dt)$. The transition is identified using new inequalities for the upper bounds for the $-dD_m^{-2}/dt$. The Navier-Stokes solutions while showing less growth in the D_m, surprisingly obey $D_m \geq D_{m+1}$ for all times.

9:44AM A23.00009 Detrended Structure-Function in Fully Developed Turbulence[1] , YONGXIANG HUANG, Shanghai Institute of Applied Mathematics and Mechanics, Shanghai University, Shanghai 200072 China — A detrended structure-function (DSF) method is proposed to extract scaling exponent by constraining the influence of large-scale structures. This is accomplished by removing a qth-order polynomial fitting within a window size τ before calculating the velocity increment. By doing so, the scale larger than τ, i.e., $r \geq \tau$, is expected to be removed. The detrending process is equivalent to a high-pass filter. We first validate the DSF by using synthesized fractional Brownian motion for mono-fractal process and lognormal process for multifractal random walk process. When applying the DSF to a turbulent velocity obtained from a high Reynolds number wind tunnel experiment with $Re_\lambda \simeq 720$, the third-order DSF demonstrates a clear inertial range with $\mathcal{B}_3(\tau) \sim \tau$ on the range $0.001 < \tau < 0.1$ sec, corresponding to a frequency range $10 < f < 1000$ Hz, which is the inertial range predicted by using the Fourier power spectrum. The directly estimated scaling exponents (resp. singularity spectrum $f(\alpha)$) agree very well with the lognormal mode with an intermittent parameter $\mu = 0.33$. Due to large-scale effects, the scaling exponents and singularity spectra provided by both the SFs and DFAs are biased.

[1]This work is sponsored by the National Natural Science Foundation of China under Grant (No. 11072139, 11032007, 11222222, 11272196 and 11202122) , "Pu Jiang" project of Shanghai (No. 12PJ1403500).

Sunday, November 24, 2013 8:00AM - 9:57AM –

Session A24 Acoustics I 319 - Daniel Bodony, University of Illinois at Urbana-Champaign

8:00AM A24.00001 The effect of sweep on forward-step noise[1] , JIN HAO, MENG WANG, University of Notre Dame — The flow and acoustic fields of swept forward-facing steps in low-Mach-number turbulent boundary layers are investigated using large-eddy simulation in combination with Lighthill's theory. The step height is 13% of the thickness of the unperturbed boundary layer at $Re_\theta = 4755$, and the step sweep angle is varied from 0° to 45° with an increment of 15°. For the same incoming flow and step height, larger sweep angles produce smaller perturbations to the boundary layer, leading to weaker surface pressure fluctuations and acoustic radiation and speedier recovery towards an equilibrium boundary layer in the downstream. With proper normalization using the free-stream velocity component normal to the step face, the sweep-independence principle is found to be approximately valid for the reattachment length, mean step-normal velocity profiles and pressure and skin friction coefficients at all sweep angles examined. In terms of surface-pressure fluctuations and radiated acoustics, sweep independence is observed for sweep angles up to 30° at relatively low frequencies.

[1]Supported by ONR Grant N00014-12-1-0553

8:13AM A24.00002 Computational Study of Shock-Associated Noise Characteristics Using LES[1] , J. LIU, A. CORRIGAN, K. KAILASANATH, Naval Research Laboratory, N. HEEB, D. MUNDAY, E. GUTMARK, University of Cincinnati — Shock-associated noise generation has been investigated by using large-eddy simulations to compute jet flows at an underexpanded jet condition with three jet temperatures. To better understand shock-associated noise generation, shock-free jets with the same fully expanded jet conditions have also been simulated. The predictions agree well with the available experimental data in both the near and far field. It is found that shock cells at this underexpanded jet condition have little impact on the jet core length and the turbulence kinetic energy distribution, whereas the heating effect has a much larger impact by increasing the initial shear-layer spreading and shortening the jet core length. Shock-associated noise dominates in the upstream direction, and the broadband peak frequencies move to higher values in downstream direction. This frequency increase is initially small in the upstream direction, but becomes much larger in the downstream direction. In addition, it is found that the heating effect increases the broadband peak frequency. Overall the heating effect increases the mixing noise and slightly reduces the shock-associated noise. This reduces the difference between the shock-containing jets and the shock-free jets as the temperature increases.

[1]This research has been sponsored by the Office of Naval Research (ONR).

8:26AM A24.00003 Noise prediction from external flows using Ffowcs-Williams and Hawkings techniques[1] , ZANE NITZKORSKI, KRISHNAN MAHESH, University of Minnesota — We investigate noise production from turbulent flow using the Ffowcs-Williams and Hawkings (FWH) acoustic analogy for general hydrodynamic flow configurations. We describe our methodology of using porous implementations of the FWH equations to calculate far-field sound from sources that are computed by either incompressible or compressible LES/DNS. We discuss a novel endcap methodology for the quadrupole source terms. The methodology allows for estimation of volumetric noise computed over a small volume as opposed to the common approach of ignoring the entire volume term while providing fewer limitations on the propagation function. We compute the noise from cylinders at various Reynolds' numbers (Re = 150, 10k, 89k) and trailing edge configurations and compare our results, base flow as well as acoustic data, to available computations and experiments.

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

8:39AM A24.00004 Vortex Noise Reductions from a Flexible Fiber Model of Owl Down , JUSTIN JAWORSKI, Lehigh University, NIGEL PEAKE, University of Cambridge — Many species of owl rely on specialized plumage to reduce their self-noise levels and enable hunting in acoustic stealth. In contrast to the leading-edge comb and compliant trailing-edge fringe attributes of owls, the aeroacoustic impact of the fluffy down material on the upper wing surface remains largely speculative as a means to eliminate aerodynamic noise across a broad range of frequencies. The down is presently idealized as a collection of independent and rigid fibers, which emerge perpendicularly from a rigid plane and are allowed to rotate under elastic restraint. Noise generation from an isolated fiber is effected by its interaction with a point vortex, whose motion is induced by the presence of the rigid half-plane and the elastically-restrained fiber. Numerical evaluations of the vortex path and acoustic signature furnish a comparison with known analytical results for stationary fibers, and results from this primitive model seek to address how aerodynamic noise could be mitigated by flexible fibers.

8:52AM A24.00005 Interaction of a turbulent boundary layer with a cavity-backed circular orifice and tonal acoustic excitation[1] , QI ZHANG, DANIEL BODONY, University of Illinois at Urbana-Champaign — Acoustic liners are effective reducers of jet exhaust and core noise and work by converting acoustic-bound energy into non-radiating, vorticity-bound energy through scattering, viscous, and non-linear processes. Modern liners are designed using highly-calibrated semi-empirical models that will not be effective for expected parameter spaces on future aircraft. The primary model limitation occurs when a turbulent boundary layer (TBL) grazes the liner; there are no physics-based methods for predicting the sound-liner interaction. We thus utilize direct numerical simulations to study the interaction of a Mach 0.5 zero pressure gradient TBL with a cavity-backed circular orifice under acoustic excitation. Acoustic field frequencies span the energy-containing range within the TBL and amplitudes range from 6 to 40 dB above the turbulent fluctuations. Impedance predictions are in agreement with NASA Langley-measured data and the simulation databases are analyzed in detail. A physics-based reduced-order model is proposed that connects the turbulence-vorticity-acoustic interaction and its accuracy and limitations are discussed.

[1]This work is funded by Aeroacoustics Research Consortium.

9:05AM A24.00006 Experimental validation of the directional sensitivity of the acoustic radiation force to particle diameter , WEIYU RAN, J.R. SAYLOR, Clemson University — A review of existing theories for the acoustic radiation force on a particle reveals a contradiction. Some theories predict that this force exhibits a change in sign at a critical particle diameter (all other parameters held constant), while other theories predict no such sign change. To ascertain which result is correct, experiments were conducted using an ultrasonic standing wave field in air. Particles were injected into this field whereupon, as expected, they migrated toward the pressure nodes of the standing wave field. The average diameter of these particles was gradually decreased. Under such conditions, the particles should either (i) continue to migrate to the pressure nodes, or (ii) migrate to the pressure anti-nodes at a critical diameter, if a change in sign of the acoustic radiation force exists. The results of these experiments will be presented, along with their implications on extant theories.

9:18AM A24.00007 Acoustic Radiation Force on a Finite-Sized Particle due to an Acoustic Field in a Viscous Compressible Fluid , SUBRAMANIAN ANNAMALAI, MANOJ PARMAR, BALACHANDAR S., Department of Mechanical and Aerospace Engineering, University of Florida, Gainesville, Florida, USA — Particles when subjected to acoustic waves experience a time-averaged second-order force known as the acoustic radiation force, which is of prime importance in the fields of microfluidics and acoustic levitation. Here, the acoustic radiation force on a rigid spherical particle in a viscous compressible medium due to progressive and standing waves is considered. The relevant length scales include: particle radius (a), acoustic wavelength (λ) and viscous penetration depth (δ). While a/λ and a/δ are arbitrary, $\delta \ll \lambda$. A farfield derivation approach has been used in determining the radiated force. Expressing the flow-field as a sum of the incident and scattered fields, an analytical expression for the force is obtained as a summation over infinite series (monopole, dipole and higher sources). These results indicate that the contributions from monopole, dipole and their cross-interaction are sufficient to describe the acoustic radiation force. Subsequently, the monopole and dipole strengths are represented in terms of the particle surface and volume averages of the incoming velocity. This generalization allows one to evaluate the radiation force for an incoming wave of any functional form. However acoustic streaming effects are neglected.

9:31AM A24.00008 Sequencing of acoustic events in the near field of subsonic jets[1] , JACQUES LEWALLE, PINQING KAN, Syracuse University — Our group has developed several pattern recognition algorithms to identify short events common to near-and far-field signals. Here we are treating far-field and near-field pressure data as well as TR-PIV (10kHz) sections through the near jet. Our algorithms are based on wavelet transforms (band-pass filtering) and cross-correlations, identifying short excerpts in the time-frequency-lag domain that contribute most to the correlations. Matching such events between multiple signal pairs exposes the sequencing of near-field activity. We consider only near-field events (NFEs) matched with a loud far-field event (FFE). The NFEs are based on the correlation of velocity, vorticity, 2-D divergence, Q-index and Kulite signals with far-field pressure. The timing of the NFEs maps out possible sequences of events related to loud coherent noise emission. Results at several subsonic Mach numbers are compared.

[1]This work is supported in part by Spectral Energies LLC, under an SBIR grant from AFRL; by a Syracuse University Graduate Fellowship; and by the Department of Mechanical and Aerospace Engineering at SU.

9:44AM A24.00009 Full-scale simulation and reduced-order modeling of a thermoacoustic engine , CARLO SCALO, Center for Turbulence Research, Stanford University, JEFF LIN, SANJIVA LELE, LAMBERTUS HESSELINK, Stanford University — We have carried out the first three-dimensional numerical simulation of a thermoacoustic Stirling heat-engine. The goal is to lay the groundwork for full-scale Navier-Stokes simulations to advance the state-of-the-art low-order modeling and design of such devices. The model adopted is a long resonator with a heat-exchanger/regenerator (HX/REG) unit on one end - the only component not directly resolved. A temperature difference across the HX/REG unit of 200 K is sufficient to initiate the thermoacoustic instability. The latter is a Lagrangian process that only intensifies acoustic waves traveling in the direction of the imposed temperature gradient. An acoustic network of traveling waves is thus obtained and compared against low-order prediction tools such as DeltaEC. Non-linear effects such as system-wide streaming flow patterns are rapidly established. These are responsible for the mean advection of hot fluid away from the HX/REG (i.e. thermal leakage). This unwanted effect is contained by the introduction of a second ambient heat-exchanger allowing for the establishment of a dynamical thermal equilibrium in the system. A limit cycle is obtained at +178 dB.

Sunday, November 24, 2013 8:00AM - 9:57AM –

Session A25 Flow Control I: Coherent Structures and Vortices 320 - Sven Schmitz, Pennsylvania State University

8:00AM A25.00001 Large-scale coherent structures in fractal-generated, axisymmetric wakes , JOVAN NEDIC, OUTI SUPPONEN, Imperial College London, BHARATHRAM GANAPATHISUBRAMANI, University of Southampton, JOHN CHRISTOS VASSILICOS, Imperial College London — The coherence and energy of large-scale structures in turbulent axisymmetric wakes are known to play a role on the drag coefficient of the body. Specifically, there is an expectation that drag can be reduced by reducing the energy of the vortex shedding. We use fractal plates which have been shown to have higher drag coefficients than square plates and disks with the same frontal area (Nedic, Ganapathisubramani & Vassilicos FDR 2013), yet show that the energy of the large-scale vortices shed from these plates is reduced by 15% to 60% compared to non-fractal plates. Fractal plates can reduce wake size and alter dissipation scalings [see DFD13-2013-000126] and the relation $C_D = C_V C_{\tilde{\epsilon}}$ between the drag coefficient and coefficients of wake volume and average turbulent dissipation rate can be used to explore consequences on drag. Furthermore, the azimuthal mode associated with the vortex shedding ($m = 1$) is still found to be dominant for all plates, however its coherence is slightly altered by the fractals, whilst mode $m = 2$ has been dramatically altered.

8:13AM A25.00002 Plasma Streamwise Vortex Generators in an Adverse Pressure Gradient , CHRISTOPHER KELLEY, THOMAS CORKE, FLINT THOMAS, University of Notre Dame — A wind tunnel experiment was conducted to compare plasma streamwise vortex generators (PSVGs) and passive vortex generators (VGs). These devices were installed on a wing section by which the angle of attack could be used to vary the streamwise pressure gradient. The experiment was performed for freestream Mach numbers 0.1–0.2. Three-dimensional velocity components were measured using a 5-hole Pitot probe in the boundary layer. These measurements were used to quantify the production of streamwise vorticity and the magnitude of the reorientation term from the vorticity transport equation. The effect of Mach number, pressure gradient, operating voltage, and electrode length was then investigated for the PSVGs. The results indicate that the PSVGs could easily outperform the passive VGs and provide a suitable alternative for flow control.

8:26AM A25.00003 Suppression of Wake Vortices Using Periodic Cross-Section Variations , A. BOUABDALLAH, Université des Sciences et de la Technologie Houari Boumediene, Algiers, Algeria, H. OUALLI, A. BENLAHNACHE, Y. MENAD, École Militaire Polytechnique, Algiers, Algeria, M. GAD-EL-HAK, Virginia Commonwealth University, Richmond, Virginia, USA — Vortices in the wake of blunt bodies are responsible for significant portion of the drag. An active flow control strategy is designed to inhibit the shedding of such vortex structures. A numerical study is conducted to investigate the effect of periodic cross-section variations on the shed vortices. We use an LES scheme with a Smagorinsky–Lilly subgrid model. The two-dimensional body sinusoidally changes its cross-section from circular to elliptic. The amplitude varies in the range of 5–100% of the nominal cylinder's diameter, and the oscillation frequency varies in the range of 0.2–10 times the cylinder's natural shedding frequency. The von Kármán vortex street is most sensitive to the cross-section variations at a Reynolds number of 3,740. At this Re, the boundary layer is subcritical, and the wake is predominately bidimensional. The flow exhibits a cascade of bifurcations identified by the shifting of the shedding mode. When the flow control strategy is optimized, as much as 65% drag reduction is achieved, which is a direct result of the shedding mechanism inhibition. An experimental validation of this result is forthcoming.

8:39AM A25.00004 Effect of the cross sectional aspect ratio on the flow past a twisted cylinder[1] , JAE HWAN JUNG, HYUN SIK YOON, Pusan National University — The cross-flow around twisted cylinders of cross sectional aspect ratio (A/B) from 1 to 2.25 is investigated at a subcritical Reynolds number (Re) of 3000 using large eddy simulation (LES). The flow past a corresponding smooth and wavy cylinder is also calculated for comparison and validation against experimental data. The effect of twisted surface assessed in terms of the mean drag and root-mean-square (RMS) value of fluctuating lift. The shear layer of the twisted cylinder covering the recirculation region is more elongated than those of the smooth and the wavy cylinder. Successively, vortex shedding of the twisted cylinder is considerably suppressed, compared with those of the smooth and the wavy cylinder. The maximum drag reduction of up to 13% compared with a smooth cylinder is obtained at a certain cross sectional aspect ratio. The fluctuating lift coefficient of the twisted cylinder is also significantly suppressed. We found that the cross sectional cross sectional aspect ratio (A/B) plays an essential role in determining the vortical structures behind the twisted cylinder which has a significant effect on the reduction of the fluctuating lift and suppression of flow-induced vibration.

[1]This work was supported by the National Research Foundation of Korea (NRF) grant funded by the Korea government (MSIP) through GCRC-SOP (No. 2011-0030013)

8:52AM A25.00005 Multi-Point Velocity Correlations in the Wake of a Three-Dimensional Bluff Body , PATRICK SHEA, MARK GLAUSER, Syracuse University — Three-dimensional bluff-bodies known as turrets are commonly used for housing optical systems on airborne platforms. These geometries generate highly turbulent wakes that decrease the performance of the optical systems and the aircraft. The current experimental study used dynamic suction in both open and closed-loop control configurations to actively control the wake turret. The experiments were carried out at a Reynolds number of 5x10^5, and the flow field was characterized using stereoscopic PIV measurements acquired in the wake of the turret. These data were processed using traditional single-point statistics which showed that the active control system was able to significantly alter the wake of the turret. Using multi-point correlations, turbulent characteristics such as the integral length scale can be calculated. For the turret wake, estimates of the integral length scales were found to be highly dependent upon the region of the flow that was evaluated, especially when comparing the shear layers to the center of the wake. With the application of the active control, the integral length scales were generally found to increase.

9:05AM A25.00006 Dynamic modeling of a turbulent axisymmetric bluff-body wake , GEORGIOS RIGAS, AIMEE MORGANS, JONATHAN MORRISON, Imperial College London — En route to chaos the stable laminar wake past axisymmetric bluff bodies undergoes two well-documented transitions by increasing the Reynolds number: a steady bifurcation of the $m = 1$ azimuthal mode followed by an unsteady bifurcation with $m = \pm 1$, the latter giving rise to periodic shedding of vortices with opposite signs, known as vortex shedding. In this study we present experimental evidence that these structures persist far from the critical points at high Reynolds numbers ($Re = 2 \times 10^5$). We show that a low-order model based on the normal form describing the codimension-two bifurcation captures accurately the dynamic behavior of the large-scale coherent structures associated with the destabilized modes, if noise is appropriately accounted for in the model. The model is validated based on simultaneous aerodynamic force measurements on the base of an axisymmetric bullet-shaped body and Time-Resolved Stereo PIV in the near wake. Finally, we extend the model to include external forcing when periodic blowing and suction is applied at the base below the point of separation.

9:18AM A25.00007 Suppressing vortex shedding behind a circular cylinder via a tangential standing wave , NANSHENG LIU, XIYUN LU, Depart. of Modern Mechanics, Univ. of Sci. & Tech. of China — Flow over a circular cylinder with a tangential standing wave imposed on the leeward surface has been numerically studied using the lattice Boltzmann method. The modulating effects of the Tangential Standing Wave (TSW) on the vortex shedding and the hydrodynamic forces have been investigated. Three vortical flow regimes in the wake have been identified, namely, Natural Vortex Shedding (NVS) dominated, NVS/TSW competing and TSW dominated regimes. Specifically, in the TSW dominated regime, alternative vortex shedding found behind an uncontrolled circular cylinder is fully suppressed so that elimination of von Karman vortex street occurs; More interestingly, an inverse von Karman vortex street leading to a net thrust is demonstrated under some certain cases of TSW control.

9:31AM A25.00008 Spanwise correlation in the wake of circular cylinder and normal plate placed inside a pipe , AMIT AGRAWAL, ARUMURU VENUGOPAL, S.V. PRABHU, Indian Institute of Technology Bombay India — The spanwise correlation of a circular cylinder and normal plate placed inside a pipe in fully developed turbulent regime is studied using hotwire anemometer. The present configuration possesses complex fluid structure interaction owing to the following features: high blockage effect, low aspect ratio of the body, upstream turbulence and interaction of axisymmetric flow with a two dimensional bluff body. Three different blockage ratios (0.14, 0.19 and 0.28) are considered. Correlation coefficient was observed to be improved with increase in blockage ratio. Compared to a circular cylinder, a normal plate possesses high correlation length. The near wall effects tend to increase the phase drift, which is reflected in low correlation coefficients. The results show that the simultaneous effect of curvature, low aspect ratio and upstream turbulence reduces the correlation coefficients significantly as compared to unconfined and confined (parallel channel) flows. The three dimensionality of vortex shedding for normal plate with a blockage ratio of 0.28 was observed to be lower compared to circular cylinder and all other blockage ratios. Low frequency modulations were found to be responsible for weak vortex shedding from a circular cylinder compared to a normal plate.

9:44AM A25.00009 Experimental Investigation of a Helicopter Rotor Hub Wake , DAVID REICH, The Pennsylvania State University Department of Aerospace Engineering, BRIAN ELBING, The Pennsylvania State University Applied Research Laboratory, SVEN SCHMITZ, The Pennsylvania State University Department of Aerospace Engineering — A scaled model of a notional helicopter rotor hub was tested in the 48" Garfield Thomas Water Tunnel at the Applied Research Laboratory Penn State. The main objectives of the experiment were to understand the spatial- and temporal content of the unsteady wake downstream of a rotor hub up to a distance corresponding to the empennage. Primary measurements were the total hub drag and velocity measurements at three nominal downstream locations. Various flow structures were identified and linked to geometric features of the hub model. The most prominent structures were two-per-revolution (hub component: scissors) and four-per-revolution (hub component: main hub arms) vortices shed by the hub. Both the two-per-revolution and four-per-revolution structures persisted far downstream of the hub, but the rate of dissipation was greater for the four-per-rev structures. This work provides a dataset for enhanced understanding of the fundamental physics underlying rotor hub flows and serves as validation data for future CFD analyses.

Sunday, November 24, 2013 8:00AM - 9:57AM –

Session A26 Reacting Flows I: Detonation 321 - J. Philip Drummond, NASA Langley

8:00AM A26.00001 What Zeldovich did not tell us about spontaneous reaction wave propagation , DAVID R. KASSOY, University of Colorado, Boulder: retired — Zeldovich (Comb. Flame, 39, 211-214, 1980) describes a conceptual model for a "spontaneous reaction wave" propagating down an initially imposed negative temperature gradient in a reactive gas. The concept is based on a sequence of constant volume thermal explosions in neighboring reactant particles of lower and lower temperature **where the interaction between the particles is neglected**. This restriction prevents nonlinear gasdynamical evolution arising from the response of a compressible gas to localized, transient thermal power addition. Gu et al. (Comb. Flame, 133, 63-74, 2003) employ a nondimensional temperature gradient parameter (actually the inverse Mach number of the propagating front) to distinguished between computationally determined diverse modes of reaction front propagation arising from spherical hot spots with imposed initial negative temperature gradients in a motionless, constant pressure gas. Two questions arise from these formulations: 1. Under what conditions can such an initial state arise? 2. Can the nondimensional parameter as used by Gu et. al. be derived from a fully compressible reactive gasdynamical formulation of their problem? These questions are addressed by employing thermomechanical principles described by Kassoy (J.Eng. Math, 68, 249 -262, 2010).

8:13AM A26.00002 Pulsed Detonation Operation of an Axial Turbine , DAVID MUNDAY, ANDREW ST. GEORGE, ROBERT DRISCOLL, EPHRAIM GUTMARK, University of Cincinnati, GAS DYNAMICS AND PROPULSION LAB TEAM — A detonation is by its nature a more thermodynamically efficient combustion mode than deflagration. Several attempts are underway to integrate detonating combustion into turbomachines in order to realize the increased efficiency available, save resources and reduce emissions. One approach to this challenge is to employ pulsed detonations as from pulsed detonation engines (PDEs) and use the pulsed outflow to drive a turbine. The difficulty with this approach is that turbines, especially the more efficient axial turbines suffer in efficiency when their inflow is pulsed. At present there is not even a commonly acknowledged turbine efficiency measure which works reasonably for a pulsed input. The present work investigates the efficiency of an axial turbine with pulsed flow. Initial tests are performed with non-combusting flow in order to study the influence of pulsation on the turbine performance. This cold flow will admit a broader range of instrumentation which can be inserted within the turbine. This allows time-resolved measure of the flow angles which have a pronounced effect on the turbine performance. Later tests with detonating inflow yield global measures and these are compared to the non-combusting results. Work sponsored by Innovative Scientific Solutions, Inc.

8:26AM A26.00003 On the development of Hydrogen-air detonations , CHRISTOPHER ROMICK, University of Notre Dame, TARIQ ASLAM, Los Alamos National Laboratory, JOSEPH POWERS, University of Notre Dame — The development and propagation of Hydrogen-air detonations is examined. An initially quiescent stoichiometric mixture at $298.15\ K$ and $1\ atm$ is initialized using a hot spot similar in character to a spark. Several two-dimensional channel widths are examined to obtain greater insight into the effect that no-slip walls have on the formation process of the detonation. To model the phenomena, the compressive, reactive Navier-Stokes equations using detailed kinetics are used with multicomponent diffusion including Soret and DuFour effects. A chemical mechanism composed of 19 reversible reactions, containing 9 species and 3 elements is used for the kinetics model. The use of detailed kinetics gives rise to multiple length scales; to predict the full richness of the unsteady behavior of a detonation, all these scales must be resolved. Resolving the finest and larger scales is accomplished using the Wavelet Adaptive Multiresolution Representation (WAMR) technique. This adaptive mesh refinement technique has a high compression ratio of the number of points needed to accurately represent the flow versus an uniform grid. The time to the initial thermal explosion is examined for the various channel widths. Additionally, the long time sustainability of the detonation is studied.

8:39AM A26.00004 Analytical and Computational Study of Flame Acceleration due to Wall Friction in Combustion Tubes and Channels , BERK DEMIRGOK, V'YACHESLAV AKKERMAN, West Virginia University — Deflagration-to-detonation transition constitutes one of the fundamental problems within the studies of reacting flows. It occurs when a subsonic flamefront accelerates, with velocity jump by several orders of magnitude. According to the Shelkin model, the key element of the process is wall friction at non-slip walls, driving a flow of the fresh pre-mixture to be non-uniform, leading to a positive flame-flow feedback and thereby flame acceleration. We perform analytical and computational study of the phenomenon, with very good agreement between them in the domain of intrinsic accuracy of the theory. Theory assumes large Reynolds number (Re) and thermal expansion as well as plane-parallel flow ahead of flamefront. Simulations are performed for complete set of combustion and hydrodynamic equations. Analytical and computational results are also validated by recent experiments on ethylene-oxygen combustion. It is proven realistic flames with a large density drop at the front accelerate in a self-sustained manner and may initiate detonation in a sufficiently long tube. Before this event, the flame shape and the velocity profile remain self-similar. Acceleration rate grows with thermal expansion in the burning process but decreases with Re related to flame propagation.

8:52AM A26.00005 Spontaneous Deflagration-to-Detonation Transition in Thermonuclear Supernovae , ALEXEI POLUDNENKO, VADIM GAMEZO, ELAINE ORAN, Naval Research Laboratory — We present the analysis of the spontaneous deflagration-to-detonation transition (DDT) in turbulent thermonuclear flames in Type Ia supernovae - explosions of degenerate white dwarf stars in binary stellar systems. We show results of first-principles numerical calculations that are used to develop and validate a subgrid-scale model for predicting the onset of DDT in full-star calculations. We also discuss detailed properties of laminar thermonuclear deflagrations for compositions and densities, at which DDT is expected to occur.

9:05AM A26.00006 Boundary Layer Effects on Ignition in a Shock-Tube System , KEVIN GROGAN, MATTHIAS IHME, Stanford University — Direct numerical simulations (DNS) of an argon-diluted hydrogen/oxygen mixture are performed to study the weak and strong ignition regimes in a shock-tube system. An adaptive mesh-refinement (AMR) algorithm is used to resolve physically relevant features such as the viscous boundary layer, the shock bifurcation region, and the ignition kernels. The simulations employ a second-order accurate, nonlinear, hyperbolic equation solver that is modified to include a finite-rate kinetic mechanism, and detailed mass, thermal, and viscous diffusion transport properties. Detailed two- and three-dimensional simulations are performed to investigate effects of viscous heating, shock bifurcation, and thermo-viscous boundary layer on the ignition behavior. The locations of the ignition kernels for various post-reflected-shock conditions are studied as well as the ignition sensitivity due to the choice of thermal boundary conditions. These detailed simulations are analyzed, and correlations between observed weak and strong ignitions are compared to the ignition criterion that was proposed by Meyer and Oppenheim.

9:18AM A26.00007 Nonlinear evolution equation for 1-D pulsating detonations with Fickett's model for reactive compressible flow, Influence of χ , ANDRE BELLERIVE, JUSTIN TANG, MATEI RADULESCU, University of Ottawa — 1-D Asymptotic analysis on Fickett's model for reactive compressible flow, i.e Burgers' equation with an added reactive term. The model's simplicity is useful to identify the mechanisms that control the detonation stability. An induction-reaction, two-step, chain-branching reaction model is used. We assume a slowed time evolution based on the particle transit through the induction zone. The equation is derived for a high activation energy and a larger exothermic reaction layer than induction layer. The evolution equation is second order in time in the shock front velocity perturbation. The equation yields both stable and unstable solutions, the unstable solutions lead to high amplitude limit-cycles. The results show the stability boundary to be the activation energy times the ratio of induction time to reaction time, $\chi < CST$, at high activation energies. For larger reaction time to induction time the stability is only dependent on the activation energy, $\alpha < CST$. The stability boundary and unstable solutions agree with numerical simulation and are consistent with previous reactive Euler models.

9:31AM A26.00008 On the role of unreacted pockets in unstable detonation waves , JONATHAN REGELE, Iowa State University — Pockets of unreacted fluid surrounded by combustion products form and react behind unstable detonation waves. It is unclear how the pockets interact with the detonation front and whether or not their reaction helps sustain detonation propagation. With the wide range of scales present in unstable detonations, unreasonable computational resources are required to perform direct numerical simulations that capture the complex interactions between diffusion, turbulence, and autoignition. In order to develop a basic understanding of what role these pockets may play, a simplified acoustic timescale analysis of unreacted pockets is performed to classify the behavior regimes. This classification is used to interpret experimental data and determine if the reaction of these pockets is isobaric and can be neglected or if compression or even shock waves are created. The generation of compression or shock waves suggests that these pockets may play a role in sustaining the detonation wave.

9:44AM A26.00009 A qualitative model for detonation with losses , ASLAN KASIMOV, LUIZ FARIA, KAUST — Burgers equation with a nonlocal forcing is capable of qualitatively reproducing many dynamical characteristics of unstable detonations. We extend previous work on the model to account for generic energy losses. A new approach is proposed for solving the nonlinear eigenvalue problem associated with the steady (or quasi-steady) detonation speed. The method eliminates difficulties associated with the sonic-point singularity and allows for easy and accurate numerical solution of the problem. We explore the role of curvature or friction in the stability of a steady detonation solution and contrast our results with analogous results in the reactive Euler equations.

Sunday, November 24, 2013 8:00AM - 9:57AM –

Session A28 Free-Surface Flows I Spirit of Pittsburgh Ballroom B/C - Julie Crockett, Bringham Young University

8:00AM A28.00001 Jet impingement and thin film breakup on a superhydrophobic surface , JULIE CROCKETT, JOSEPH PRINCE, DANIEL MAYNES, Brigham Young University — A vertical laminar jet impinging on a horizontal surface spreads out in a thin film on the surface. If the surface is hydrophobic, and a downstream depth is not maintained the film will breakup into droplets. This occurs where the jet's outward radial momentum is balanced by the inward surface tension force of the advancing film. An analytical model has been created to estimate this location. All surfaces explored are hydrophobic or superhydrophobic (SH), where the SH surfaces exhibit an apparent slip at the surface. For SH surfaces with random micropatterning, the slip on the surface is uniform in all directions and droplet breakup occurs in a circular pattern. When alternating rib/cavity microstructures are used to create a SH surface the slip varies as a function azimuth resulting in an elliptically shaped breakup. The location of breakup for multiple surfaces over a range of jet Weber numbers and realistic slip length values is determined. Results show the breakup radius increases with increasing Weber number and slip length. The eccentricity of the breakup ellipse for the rib/cavity SH structures increases with increasing Weber number and slip length as well. The model results compare well to experimental measurements.

8:13AM A28.00002 Absolute and convective instabilities in film flow over inclined topography , DMITRI TSELUIKO, Department of Mathematical Sciences, Loughborough University, MARK BLYTH, School of Mathematics, University of East Anglia, DEMETRIOS PAPAGEORGIOU, Department of Mathematics, Imperial College London — The stability of a liquid film flowing under gravity down an inclined wall with periodic corrugations is analyzed. A long-wave equation valid at near-critical Reynolds numbers is used to study the film dynamics. Steady solution branches are computed including subharmonic branches, for which the period of the free surface is an integer multiple of the wall period, and the existence of quasi-periodic branches is demonstrated. Stability analysis of steady periodic solutions shows that under certain conditions, and depending on the wall period, the flow may be convectively unstable for small wall amplitudes but undergo transition to absolute instability as the wall amplitude increases. The predictions of the linear theory are corroborated by time-dependent simulations of the model equation.

8:26AM A28.00003 The miscible two-fluid flow down an inclined plane: Linear stability analysis , R. USHA, Department of Mathematics, IIT Madras, Chennai-600036, India, RAMA GOVINDARAJAN, TIFR centre for Interdisciplinary Sciences, TATA Institute of Fundamental Research, Hyderabad-500075, India, OUTI TAMMISOLA, Department of Engineering, University of Cambridge, Trumpington Street, Cambridge, CB2 1PZ UK — The linear stability of a miscible two-layer free-surface flow of varying viscosity, down an inclined substrate is examined. We show that the stability characteristics are different from both immiscible two-layer flows and continuously stratified flows. A new instability mode, namely overlap mode, absent in either limiting case, arises when the critical layer of the disturbance overlaps the viscosity-stratified layer. At moderate miscibility, the configuration with less viscous fluid adjacent to the inclined plane is most stabilizing. This is also contrast with the limiting cases, in which the lubrication configuration is always destabilizing. The co-existence of several growing overlap modes, the usual surface mode and a Tollmien-Schlichting mode are observed and this presents interesting new possibilities for nonlinear breakdown.

8:39AM A28.00004 Thin film flow down a porous substrate in the presence of a soluble surfactant: Linear stability analysis , YADAV ANJALAIAH, R. USHA, Department of mathematics, IIT Madras, Chennai-600036, India — The linear stability of a thin film flowing down an inclined porous substrate in the presence of soluble surfactants is investigated. A surfactant model in which the surfactant has affinity only for the liquid-gas and not for the liquid-solid interface, and is contained in the bulk only as a monomer is considered. The adsorption-desorption kinetics of the surfactant at the liquid-gas interface is accounted for. An Orr-Sommerfeld eigenvalue problem is formulated and is solved analytically in the limit of long-wave perturbations and numerically for arbitrary wave-length using Spectral-Tau collocation method. The effects of solubility of the surfactant, the characteristics of the porous medium and adsorption-desorption kinetics are examined. The results reveal the stabilizing effect of soluble surfactant on the flow system. It is possible to either stabilize or destabilize the flow system by appropriately choosing the characteristics of the porous medium. The presence of soluble surfactants is shown to be more effective in stabilizing the flow system than that of insoluble surfactants.

8:52AM A28.00005 Recoil of a liquid filament: escape of the pinch-off by creation of a vortex ring , JEROME HOEPFFNER, GOUNSETI PARE, Université Pierre et Marie Curie — A liquid filament recoils under the effect of its surface tension. It may recoil to one sphere: the geometrical shape with lowest surface, or otherwise segment to several pieces which individually will recoil to spheres. This experiment is classical and its exploration is fundamental to understanding how liquid volumes relax. In this talk, we uncover a mechanism involving the creation of a vortex ring which plays a central role in escaping the segmentation.

9:05AM A28.00006 Spinning hydraulic jump , HAMID ABDERRAHMANE, ASLAN KASIMOV, KAUST — We report an experimental observation of a new symmetry breaking of circular hydraulic jump into a self-organized structure that consists of a spinning polygonal jump and logarithmic-spiral waves of fluid elevation downstream. The waves are strikingly similar to spiral density waves in galaxies. The fluid flow exhibits counterparts of salient morphological features of galactic flows, in particular the outflow from the center, jets, circum-nuclear rings, gas inflows toward the galactic center, and vortices. The hydrodynamic instability revealed here may have a counterpart that plays a role in the formation and sustainability of spiral arms in galaxies.

9:18AM A28.00007 Linear and weakly nonlinear analysis of the rotating polygon instability , JEROME MOUGEL, DAVID FABRE, Institut de Mecanique des Fluides de Toulouse, TOMAS BOHR, Technical University of Denmark — In this talk we will present new analytic results about the polygonal instability obtained in a cylindrical container with rotating bottom [G. H. Vatistas, J. Fluid. Mech, **217**, 241, (1990), Jansson et al., Phys. Rev. Lett, **96**, 174502, (2006)]. In a recent study we showed that this spectacular instability can be explained as a result of wave interaction by introducing a simplified model that allows analytical predictions [L. Tophøj et al., Phys. Rev. Lett, **110**, 194502, (2013)]. Instability maps of the global stability analysis will be presented here, as well as results of the weakly nonlinear analysis performed on the simple model which lead to the amplitude equations of the resonating free surface waves.

9:31AM A28.00008 Meniscus Stability in Rotating Systems[1] , YVONNE REICHEL, MICHAEL DREYER, ZARM University of Bremen, Germany — In this study, the stability of free surfaces of fluid between two rotating coaxial, circular disks is examined. Radially mounted baffles are used to form menisci of equal size. To the center of the upper disk, a tube is connected in which a separate meniscus is formed. Assuming solid-body rotation and ignoring dynamic effects, it is observed that the free surfaces between the disks fail to remain stable once the rotation speed exceeds a critical value. In other words, Rayleigh-Taylor instability ensues when the capillary forces fail to balance centrifugal forces. Dimensionless critical rotation speeds are studied by means of the Surface Evolver via SE-FIT for varied number of baffles, the normalized distance between the disks, and the normalized central tube radius. Drop tower tests are performed to confirm some of the numerical results. The computation also reveals that there are different modes of instability as a function of the relevant parameters.

[1]This study was funded by the space agency of the German Aerospace Center with resources of the Federal Ministry of Economics and Technology on the basis of a resolution of the German Bundestag under grant number 50 RL 1320.

9:44AM A28.00009 Hydraulic jumps and contact lines formed by jet impact on an incline , LAURENT LIMAT, ALEXIS DUCHESNE, REMY HERBAUT, LUC LEBON, Matiere et Systemes Complexes (MSC), UMR 7057 of CNRS and Univ. Paris Diderot, Paris, France — We have investigated the shape and stability of hydraulic jumps formed on an inclined plate, around a jet under normal impingement. We have explored three different wetting conditions: total wetting, partial wetting and super-hydrophobicity. In the first case, a strong departure to axisymmetry of the shape is observed, with often disappearance of the lower part of the jump. One also observes the formation of an effective, curved, static contact line around the jump, with a similar horse-shoe structure. Surprisingly, the effective jump radii defined in the directions normal and parallel to the in plane gravity follow quite well Bohr et al scaling, initially proposed for a horizontal, axisymetric jump, but with prefactors dependant on the plate slope. In the partial wetting case, the coupling between the jump and the contact line makes things more complex and Bohr' scaling seems to hold only at large plate slope. In the super hydrophobic case, the structure is strongly axisymmetrical, and reminiscent of sheet atomization. The sheet radius is governed by a balance between surface tension and momentum, itself moderated by the viscous friction on the plate.

Sunday, November 24, 2013 8:00AM - 9:57AM –

Session A30 Instability: Richtmyer-Meshkov 408 - Davesh Ranjan, Texas A&M University

8:00AM A30.00001 Large Eddy Simulation Requirements for the Richtmyer-Meshkov Instability , BRITTON OLSON, JEFF GREENOUGH, Lawrence Livermore National Laboratory — The shock induced mixing of two gases separated by a perturbed interface is investigated through Large Eddy Simulation and Direct Numerical Simulation. At coarse resolutions, the effects of numerical dissipation outweigh those of physical dissipation on the entrainment and mixing process of the Richtmyer-Meshkov Instability. Decreasing the Reynolds of the flow while increasing the grid resolution largely mitigates the relative numerical dissipation but is often unachievable for realistic flows. A model for an effective viscosity is proposed which allows for an *a posteriori* analysis of the simulation data that is agnostic to the LES model, numerics and the physical Reynolds number of the simulation. An analogous approximation for an effective species diffusivity is also presented. This framework can then be used to estimate the effective Reynolds number and Schmidt number of future simulations and elucidate the impact of numerical dissipation on the mixing process for an arbitrary numerical method.

8:13AM A30.00002 Experimental Parametric Study of the Inclined Interface Richtmyer-Meshkov Instability , JACOB MCFARLAND, SKYLAR CREEL, DAVID REILLY, CHRISTOPHER MCDONALD, Texas A&M University, JEFFREY GREENOUGH, Lawrence Livermore National Laboratory, DEVESH RANJAN, Texas A&M University — Experiments performed in the Texas A&M shock tube facility will be presented which explore the effect of incident shock strength, Atwood number, and inclination angle on the development of the Richtmyer-Meshkov instability. Experiments with a range of Atwood numbers ($\sim$0.23 to 0.85) and high interface inclination angles ($>45^\circ$) at moderate incident shock Mach numbers ($\sim$1.55 and 1.9) will be presented. Qualitative results will be presented using Mie scattering images obtained from early to late times before reshock. Quantitative results such as the interface mixing width growth rate, and vorticity deposition will be presented for select cases. Experimental results will also be compared with simulation results from the Lawrence Livermore National Laboratory's ALE hydrodynamics code, ARES.

8:26AM A30.00003 Experimental Study of the Richtmyer-Meshkov Instability on a Coupled Multimode and Inclined Interface Perturbation[1] , DAVID REILLY, SKYLAR CREEL, JACOB MCFARLAND, Texas A&M University, JATIN MITRUKA, Indian Institute of Technology Kanpur, CHRISTOPHER MCDONALD, DEVESH RANJAN, Texas A&M University — The inclined shock tube in the Texas A&M Shock Tube and Advanced Mixing Laboratory was used to study the effect of small amplitude, long wavelength multimode perturbations imposed on the inclined interface initial condition of the Richtmyer-Meshkov instability. The inclined interface is essentially a long wavelength, extremely large amplitude perturbation. Images of the shocked flow-field were captured with the angle of the shock tube with respect to the horizontal at 60° ($\eta/\lambda = \sqrt{3}/6$). The modal content of the initial conditions was determined by taking the Fourier decomposition of the interface. This work is a proof of concept for creating a coupled multimode and inclined interface. Work that is currently underway will investigate the effect of these initial conditions on intermediate and late-time mixing as well as the transition to turbulence before reshock by using qualitative comparisons of Mie scattering images, mixing width measurements, and circulation from Particle Image Velocimetry (PIV).

[1]This research was funded by the Air Force Office of Scientific Research Young Investigator Research Program (AFOSR-YIP) grant.

8:39AM A30.00004 Inclined Interface Richtmyer-Meshkov Instability: Reshock Study[1] , SKYLAR CREEL, JACOB MCFARLAND, DAVID REILLY, CHRIS MCDONALD, Texas A&M University, SHANAE SMITH, Sam Houston State University, DEVESH RANJAN, Texas A&M University — Experimental work performed in the Texas A&M University Shock Tube and Advanced Mixing Lab will be presented focusing on the effort to drive the Richtmyer-Meshkov instability to a turbulent state through the use of reshock. Experiments presented will feature a range of Atwood numbers ($\sim$0.23 to 0.67) at an inclination angle of 60°, Mach numbers of $\sim$1.55 and $\sim$1.91 and mulitiple reshock interaction times. Experiments will qualitatively detail the effect of reshock interaction time on the developing instability through Mie scattering images. Velocity fields will be acquired through the use of particle image velocimetry (PIV). Quantitative measurements of vorticity, using velocity fields, and mixing width growth rates, using Mie scattering images, of the reshocked flow will be compared to their pre-reshock values. Comparison will provide information on the effect of reshock on the level of turbulence in the flow.

[1]National Science Foundation CAREER Award

8:52AM A30.00005 ABSTRACT WITHDRAWN –

9:05AM A30.00006 Three-dimensional features of shock-driven mixing flow[1] , DELL OLMSTEAD, PETER VOROBIEFF, CLINT CORBIN, TENNILLE BERNARD, PATRICK WAYNE, GARRETT KUEHNER, C. RANDALL TRUMAN, The University of New Mexico — Richtmyer-Meshkov instability (RMI) is created by passing an oblique shock wave across a cylindrical column of heavy gas (sulfur hexafluoride SF_6) in air at Mach numbers ranging from 1.2 to 2.0. These initial conditions are inherently three-dimensional, unlike nominally two-dimensional conditions used in many earlier works. To capture the development of the RMI, Planar Laser Induced Fluorescence (PLIF) images were obtained in multiple planes along and across the RMI-perturbed column. The oblique shockwave is obtained in a shock tube inclined up to 30 degrees with the horizontal and using gravity-driven (vertical) flow to form the SF6 cylinder. The development of RMI for a cylindrical interface subjected to a normal shockwave is also documented. The main subject of the investigation is the role of the angle between the cylinder and the shock front in the formation and evolution of the three-dimensional features in the flow. Experiments also show that consideration must be given to the effects of the walls of the shock tube and especially of the holes in the walls used to form the heavy-gas column.

[1]This research is supported by NNSA (US National Nuclear Security Agency).

9:18AM A30.00007 Dependence of Single-Interface Richtmyer-Meshkov Mixing on Mach Number using Simultaneous PIV and PLIF Measurements , BRANDON M. WILSON, RICARDO MEJIA-ALVAREZ, KATHY P. PRESTRIDGE, Los Alamos National Laboratory — Richtmyer-Meshkov mixing is dependent upon initial interface perturbations, incident shock Mach number, Atwood number, and other fluid properties. The correlation between turbulence quantities and mixing parameters with these properties is not well-understood. The Vertical Shock Tube (VST) at Los Alamos National Lab is designed to measure turbulence and mixing from the Taylor micro-scale to the largest scales (mix width). We use simultaneous velocity (PIV) and density (PLIF) diagnostics to understand the effects of incident Mach number on statistically-invariant, multimode perturbations of an air-SF6 interface (Atwood number corrected for acetone diagnostic is A =0.57). We quantify Ma effects on mixing at both large and small scales by measuring the time evolution of various mixing parameters ($e.g.$ mixing width, Favre-averaged Reynolds stresses, and vorticity), and we compare these results to previous studies. Late mixing after the first shock resembles a turbulent flow, and we examine the nature of the turbulence and condition of turbulent transition.

9:31AM A30.00008 Simultaneous PIV/PLIF measurements of Richtmyer-Meshkov Instabilities from single- and multi-mode perturbed interfaces , RICARDO MEJIA-ALVAREZ, BRANDON WILSON, KATHY PRESTRIDGE, Los Alamos National Laboratory, EXTREME FLUIDS TEAM — To support validation of RANS and LES codes for single-interface Richtmyer-Meshkov mixing, the Extreme Fluids Team at Los Alamos National Laboratory commissioned a Vertical Shock Tube. This facility has the capability of generating statistically stationary single- and multi-mode spatial perturbations on the fluid interface prior to shock-interface interaction. The present study focuses on comparing the evolution of shock-driven mixing under two different spatial perturbation conditions after interacting with a M=1.2 shock wave. High resolution simultaneous PIV and PLIF are used for capturing 2D instantaneous realizations of velocity and density at different stages of the evolving interface. Multiple realizations of the flow at each one of these evolution stages are obtained to characterize the flow statistically. Also, a modal analysis via Singular Value Decomposition is performed on the density and velocity fields to elucidate the role of initial flow scales content on the transition to turbulent mixing.

9:44AM A30.00009 Simultaneous Concentration and Velocity Field Measurements in a Shock-accelerated Mixing Layer[1] , DANIEL REESE, JASON OAKLEY, University of Wisconsin-Madison, CHRIS WEBER, Lawrence Livermore National Laboratory, DAVID ROTHAMER, JOSE NAVARRO, RICCARDO BONAZZA, University of Wisconsin-Madison — The Richtmyer-Meshkov instability (RMI) is experimentally investigated at the Wisconsin Shock Tube Laboratory. Simultaneous concentration and velocity field measurements from the mixing layer of experimental RMI images are obtained through the application of the Advection-Corrected Correlation Image Velocimetry (ACCIV) technique. A statistically repeatable broadband initial condition is created by first setting up a gravitationally stable stagnation plane of helium+acetone over argon and then injecting the gases horizontally at the interface to create a shear layer. The shear layer is then accelerated by a Mach 2.2 planar shock wave that causes the growth of any perturbations present at the interface, and time-separated image pair data of the mixing layer are obtained using planar laser induced fluorescence (PLIF). The image pair is corrected to show relative acetone concentration, and is then used as input to the ACCIV algorithm to obtain velocity field results. These velocity field measurements are compared with those obtained from numerical simulations. Turbulent kinetic energy spectra are compared with particle imaging velocimetry (PIV) and simulation results to validate regions of applicability.

[1]We wish to thank the Department of Energy National Nuclear Security Administration for supporting this work.

Sunday, November 24, 2013 8:00AM - 9:44AM –

Session A31 Porous Media Flows I - General 402 - Daniel Lester, CSIRO

8:00AM A31.00001 A comprehensive study of the lift generation in soft porous media under rapid compression , QIANHONG WU, Villanova University, RUNGUN NATHAN, Penn State Berks, ROBERT CRAWFORD, Villanova University, VU CBMSS TEAM — Lift generation in soft porous media under rapid compaction is a new concept for porous media flow, which has broad applications in biological systems, squeeze damping, and soft lubrication, etc. Previous studies on this topic share a common feature of neglecting the lift contribution of the undeformed porous structures surrounding the compressed porous media, thus deviated from real applications. Herein we report a comprehensive experimental and theoretical approach to treat this shortcoming. A soft, polyester, fibrous, porous material with specified micro-structure, porosity and permeability was dynamically compressed by a loaded piston in a porous-walled cylinder-piston apparatus. Pore air was forced out radially either directly to the ambient ("unconfined" case) or to the surrounding undeformed porous media ("confined" case). Detailed pressure measurements indicate that the air lifting force underneath the piston was enhanced by 25% to 30% for the "confined" case as compared to the "unconfined" case. A consolidation theory was developed to characterize this process, which shows very good agreements with the experimental data. This study significantly improves our understanding of the dynamic response of soft porous media under rapid compression.

8:13AM A31.00002 Characterisation of flux sensitivity to uncertainty in porous media , A.J. EVANS, BP Institute, University of Cambridge, C.P. CAULFIELD, BP Institute & DAMTP, University of Cambridge, ANDREW W. WOODS, BP Institute, University of Cambridge — Natural porous media are typically heterogeneous on a range of length scales, and this leads to the challenge of defining effective medium properties with which estimates of fluid flow may be calculated. We develop a method to explore how the possible variability in the effective properties may impact estimates of the flow of a single phase fluid through a porous media. We use this method to explore how flow predictions are sensitive to uncertainty in the permeability field, and we develop the approach to explore how data may be used to reduce such uncertainty.

8:26AM A31.00003 Comparison of Experimental and Computational Methods in the Study of Flow in Porous Media[1] , JAMES LIBURDY, Oregon State University, VISHAL PATIL, Malvern Instruments, JUSTIN FINN, University of Liverpool, SOURABH APTE, Oregon State University — Both experimental and computational methods applied to the study of porous media flows are challenging due to the complex multi-phase geometry and ability to resolve scales over a reasonably large domain. This study compares experimentally obtained results based on refractive index matching of detailed velocity field vectors with those obtained using direct numerical simulations to evaluate both methods for consistency. Data were obtained in a randomly packed bed using uniformly sized spherical particles. Challenges associated with proper experimental methods including refractive index matching errors, magnification uncertainties, and the identification of the proper geometry are discussed. In addition the DNS challenges associated with, matching the geometry, grid resolution particularly near solid contact points , and proper boundary conditions are presented. Results are compared, with attention paid to identifying the relative uncertainty limitations based on the experimental and computational parameters for steady flow conditions within the bed.

[1]Support for this work was provided by NSF, Particulate and Multiphase Processing, grant 0933857

8:39AM A31.00004 Micro-scale flow simulation and colloid transport modeling in saturated porous media[1] , QUEMING QIU, 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. This talk focuses on the representation of micro-scale flow and colloid-grain surface interactions in a computational approach with 3D porous media packed with glass beads. A corresponding 2D porous media is also developed to save some computational efforts. After solving the flow field with the Lattice Boltzmann method, a Lagrangian colloid tracking model is used to study the dynamics of colloidal particles considering Brownian force, hydrodynamic forces, and physicochemical forces. The attachment efficiency at favorable condition in our 3D model is compared with experimental data and also the efficiency predicted from other research group with different models. Under the unfavorable condition, the modeling and analysis of colloid transport will explore the effects of solution ionic strength on colloid reversible retention in both 2D and 3D models. To speed up our colloid tracking modeling, parallel implementation using Message Passing Interface (MPI) is performed and the related complexity analysis and scalability results will also be presented.

[1]This work is partially supported by NSF.

8:52AM A31.00005 Flow Intermittency, Dispersion, and Correlated Continuous Time Random Walks in Porous Media , PIETRO DE ANNA, MIT, Cambridge (MA, USA), TANGUY LE BORGNE, Universite de Rennes 1, (France), MARCO DENTZ, IDAEA-CSIC, Barcelona (Spain), ALEXANDRE TARTAKOVSKY, PNNL, Richland (WA, USA), DIOGO BOLSTER, University of Notre Dame, South Bend (IN, USA), PHILIPPE DAVY, Universite de Rennes 1, (France) — We study the intermittency of fluid velocities in porous media and its relation to anomalous dispersion. The complexity of the pore scale flow arises from the heterogeneous medium structure that induces non-Gaussian velocity distributions, which can lead to a persistent non-Fickian dispersion. Lagrangian velocities measured at equidistant points along streamlines are shown to form a spatial Markov process. As a consequence of this remarkable property, the dispersion of fluid particles can be described by a continuous time random walk with correlated temporal increments. This new dynamical picture of intermittency provides a direct link between the microscale flow, its intermittent properties, and non-Fickian dispersion.

9:05AM A31.00006 SPH numerical simulation of fluid flow through a porous media[1] , JAIME KLAPP-ESCRIBANO, Cinvestav, Mexico and ININ, Mexico, ESTELA MAYORAL-VILLA, MARIO ALBERTO RODRIGUEZ-MEZA, EDUARDO DE LA CRUZ-SANCHEZ, ININ, Mexico, LEONARDO DI G SIGALOTTI, IVIC, Venezuela, ININ-ABACUS COLLABORATION, IVIC COLLABORATION — We have tested an improved a method for 3D SPH simulations of fluid flow through a porous media using an implementation of this method with the Dual-Physics code. This improvement makes it possible to simulate many particles (of the order of several million) in reasonable computer times because its execution on GPUs processors makes it possible to reduce considerably the simulation cost for large systems. Modifications in the initial configuration have been implemented in order to simulate different arrays and geometries for the porous media. The basic tests were reproduced and the performance was analyzed. Our 3D simulations of fluid flow through a saturated homogeneous porous media shows a discharge velocity proportional to the hydraulic gradient reproducing Darcy's law at small body forces. The results are comparable with values obtained in previous work and published in the literature for simulations of flow through periodic porous media. Our simulations for a non saturated porous media produce adequate qualitative results showing that a non steady state is generated. The relaxation time for these systems were obtained.

[1]Work partially supported by Cinvestav-ABACUS, CONACyT grant EDOMEX-2011-C01-165873.

9:18AM A31.00007 Numerical Study of Usage Efficiency of Multistage Filters on Mineral Leaching Process , MEDET INKARBEKOV, ALIBEK KULJABEKOV, KARLYGASH ALIBAYEVA, AIDARKHAN KALTAYEV, Al-Farabi Kazakh National University — The numerical study of the usage efficiency of the multistage filters setting technology is carried out on the basis of mathematical simulation. And its application on in-situ mineral leaching process is considered. So long as mineral bearing sandstone in deposit mostly is separated by interbedded layers of sands and clays, it's expedient to use multistage filters setting technology at the mineral extraction. A comparison of the extraction degree at single and multistage filters is implemented. The results of calculations show that the distribution of flow (inflow) on well height is not uniform. In the calculations the well accepted as high-permeability channel, depending on the construction of the filter. Obtained results for a multistage filters setting qualitatively conform to the experimental findings. Wellbore is considered as a surface with a constant reduced pressure in the bottomhole formation zone. But such assumption does not show a qualitative picture of the fluid flow in the bottomhole zone [Brovin K.G., Grabovnikov V.A., 1997]. To construct an accurate mathematical model it's necessary to use Navier-Stokes equation for the interior of a vertical wellbore, and the filtration law for modeling the filtration in the reservoir. Strictly speaking, it would have had to sew two laws on the contact surface of a rock and filter. Such review requires enormous computing, as far as computational grid must be sufficiently thick to cover the interior of the wellbore.

9:31AM A31.00008 Dynamics of non- Newtonian fluid flow in porous media , SHIMA PARSA , HARRY CHIANG, SUJIT DATTA, DAVID WEITZ, Harvard University — We study the flow of a shear thinning polymer solution within a three dimensional model porous medium made of closely packed glass beads. The polymer solution is index matched with the glass so using confocal microscopy we are able to probe the dynamics of the flow at the pore scale in the bulk of the medium. We measure the fluid velocity field in the porous medium with particle image velocimetry technique. The probability distribution of the measured velocities has an exponential tail indicating the presence of large velocities compared to the average imposed velocity. The distribution of velocities also shows correlations with the pore size. We also compare the dynamics of the flow with the case of an additional immiscible fluid trapped within the medium. The probability distribution of velocities in the presence of residual trapped oil has a wider distribution as a result of the enhanced complexity of the medium.

Sunday, November 24, 2013 8:00AM - 9:44AM –

Session A32 Surface Tension Effects I: Particles and Structure at the Interface 403 - Steve Wereley, Purdue University

8:00AM A32.00001 An asymmetric Cheerio: Torque on objects floating on a liquid interface , MICHAEL MILLER, KHOI NGUYEN, SHREYAS MANDRE, Brown University — Objects suspended on a fluid interface by surface tension are subject to attractive and repulsive forces towards each other. When symmetry is broken, a capillary torque acts to rotate the objects into an equilibrium orientation. We employ optical refraction from the meniscus to obtain the shape of the liquid surface, and use the shape to calculate the torque. Through application of an analytical asymptotic analysis at a polygonal object's vertices, supported by empirical results, we gain an understanding of the rotational effects of sharp corners in a suspended object's contact line.

8:13AM A32.00002 Designing spherical patchy particles for optimum surface activity at liquid-fluid interfaces , HOSSEIN REZVANTALAB, ALI HASHEMI, SHAHAB SHOJAEI-ZADEH, Rutgers, The State University of New Jersey — Adsorption of spherical patchy particles to a flat liquid-fluid interface is investigated. Chemical heterogeneity in form of patches with different number and size can be introduced on the surface of a homogeneous particle to induce an amphiphilic character. Compared to homogeneous particles, amphiphilic particles show stronger adhesion to the interface and thus can be used as effective interface stabilizers. Using the concept of detachment energy for a particle at an interface, we analytically developed a criterion to design single-patch particles with maximum surface activity. The method is then extended to include particles with two symmetric patches. The energy landscapes reveal the existence of two local minima for such double-patch particles when the patches are oriented either parallel or normal to the interface. We evaluate the effectiveness of introducing the second patch on the particle as a function of its size and wettability. Addition of a second symmetric patch can enhance the surface activity, compared to a single – patch particle, provided that the patch groups are relatively small, each with close affinity to the fluid phases. On the other hand, single-patch particles are more surface active for highly amphiphilic cases.

8:26AM A32.00003 The Effect of Contact Angle on the Orientation, Stability and Assembly of Dense Floating Cubes , JONATHAN ROTHSTEIN, MICHAEL DONNELL, ROBERT DANIELLO, University of Massachusetts - Amherst — In this talk, the effect of contact angle, density and size on the orientation, stability and assembly of floating cubes will be presented. All the cubes tested were denser than water. Floatation occurred as a result of capillary stresses induced by deformation of the air water interface. The cubes were observed to float in one of three primary orientations depending on contact angle: edge up, vertex up and face up. Measurements net force on the cubes showed that the maximum capillary forces were always experienced for the face up orientation. However, when floatation was possible in the vertex up orientation, it was found to be the most stable cube orientation because it had the lowest center of gravity. A series of theoretical predictions were performed for the cubes floating in each of the three primary orientations to calculate the net force on the cube. The theoretical predictions were found to match the experimental measurements well. The assembly of cubes floating face up and vertex up were also studied for assemblies of two, three and many cubes. Cubes floating face up were found to assemble face to face and form regular checkerboard arrays with no free interface between cubes. Cubes floating vertex up cubes were found to assemble in a variety of different arrangements including edge to edge, vertex to vertex, face to face and vertex to face with the most probably assembly being edge to edge. Large numbers of vertex up cubes were found to pack with a distribution of orientations and alignment.

8:39AM A32.00004 Estimation of Forces between Objects in the Cheerios Effect , KHOI NGUYEN, MICHAEL MILLER, SHREYAS MANDRE, Brown University — We develop an analysis and method to estimate the attractive forces due to the capillary interactions between nearby interfacial objects, often dubbed as the Cheerios effect. The method involve mapping the 3-dimensional surface deformation surrounding a floating object and estimating the force directly from the gradient field. The surface deformation is obtained by correlating the image of a random dot background with its refraction from the liquid surface. The scaling of attractive force as a function of distance for objects of arbitrary shape is directly obtainable from a series of gradient fields. We specialize to the case of objects with sharp corners and observe that force scales exponentially with distance on the order of a capillary length. Furthermore, this optical method is potentially instrumental in studying colloidal self-assembly because it can be implemented in real time in a dynamically changing array of floating objects.

8:52AM A32.00005 Inter-particle interactions and assembly of ellipsoidal Janus particles at liquid interfaces , HOSSEIN REZVANTALAB, STEPHEN ROWE, SHAHAB SHOJAEI-ZADEH, Rutgers, The state University of New Jersey — We study the capillary-induced interactions between ellipsoidal Janus particles adsorbed at flat liquid-fluid interfaces. In contrast to spherical particles, isolated Janus ellipsoids with a large aspect ratio or a small difference in the wettability of the two regions tend to tilt at their equilibrium orientation. The interface around such tilted particles deforms and their overlap results in capillary interactions between neighboring particles. These interactions are quantified through minimization of interfacial energy variation as a function of the separation distance between the particles. We show that Janus ellipsoids prefer to align side-by-side at the interface. We also evaluate the role of particle aspect ratio and the degree of amphiphilicity between its two regions on the inter-particle capillary forces and torques. For particles of equal surface area, the energy profiles are independent of the aspect ratio, while increasing the amphiphilicity results in an enhancement in the capillary force in-contact and the torque inducing the side-by-side configuration. This indicates that Janus ellipsoids exhibit stronger interactions compared to their homogeneous counterparts.

9:05AM A32.00006 Induced phase transitions of nanoparticle-stabilized emulsions , STEFAN FRIJTERS, FLORIAN GÜNTHER, JENS HARTING, Eindhoven University of Technology — Nanoparticles can stabilize fluid-fluid interfaces over long timescales and are nowadays commonly used, e.g. in emulsions. However, their fundamental properties are as of yet poorly understood. Nanoparticle-stabilized emulsions can exhibit different phases, such as Pickering emulsions or bijels, which can be characterized by their different topologies and rheology. We investigate the effect of various initial conditions on random mixtures of two fluids and nanoparticles - in particular, the final state these systems will reach. For this, we use the well-established 3D lattice Boltzmann method, extended to allow for the added nanoparticles. After the evolution of the emulsions has stopped, we induce transitions from one state to another by gradually changing the wettability of the nanoparticles over time. This changes the preferential local curvature of the interfaces, which strongly affects the global state. We observe strong hysteresis effects because of the energy barrier presented by the necessary massive reordering of the particles. Being able to change emulsion states *in situ* has potential application possibilities in filtering technology, or creating particle scaffolds.

9:18AM A32.00007 Particle shedding from coated magnetic microbubbles , CHON U. CHAN, School of Physical and Mathematical Science, Nanyang Technological University, YU GAO, CHENJIE XU, School of Chemical and Biomedical Engineering, Nanyang Technological University, MANISH ARORA, CLAUS-DIETER OHL, School of Physical and Mathematical Science, Nanyang Technological University — Nanoparticle-coated microbubbles have found applications for diagnostic imaging as well as drug delivery, yet the release of shell material due to ultrasonic excitation has not been studied in detail. We find that particles are ejected from the shell if the bubbles are driven resonantly with ultrasound. The bubble oscillation and the release of the particles are observed with high-speed photography while floating in microfluidic channels. The nanoparticles are shed when the bubble wall acceleration exceeds a threshold. When bubbles are excited into shape oscillations the particle shedding is observed at the velocity antinodes of the bubble surface. Particle release and transport is modelled with a force balance, considering inertia of the particle, the oscillatory fluid flow created by bubble oscillation, and the viscous force acting on the particle. We also demonstrate the ability to control the location of microbubbles in liquid using a diverging magnetic field and an optical feed-back loop. Then the magnetic microbubbles can be stabilized and translated in a three dimensional bubble trap.

9:31AM A32.00008 Visualization and minimization of clustering of micro-pillars and walls due to liquid film evaporation[1] , TAE-HONG KIM, JUNGCHUL KIM, HO-YOUNG KIM, Seoul National University — The spin drying, in which a rinsing liquid deposited on a wafer is rapidly dried by wafer spinning, is an essential step in the semiconductor manufacturing process. While the liquid evaporates, its meniscus straddles neighboring submicron-size patterns such as pillars and walls. Then the capillary effects that pull the patterns together may lead to direct contact of the patterns, which is often referred to as pattern leaning. This poses a problem becoming more and more serious as the pattern size shrinks and the aspect ratio of the patterns increases. While the clustering behavior of high-aspect-ratio micro- and nanopillars was investigated before, a technical strategy to prevent such clustering has been pursed in industrial practices without being supported by the recently established theory of elastocapillarity. Here we visualize the clustering behavior of polymer micropatterns with the evaporation of liquid film while varying the sizes and temperature of the micropatterns. We find a critical role of substrate temperature in preventing the leaning of the patterns via changing the evaporation rate and behavior of the liquid film. Also, we construct a regime map that guides us to find a process condition to avoid pattern leaning in semiconductor manufacturing.

[1]This work was supported by the National Research Foundation of Korea (grant no. 2012-008023).

Sunday, November 24, 2013 8:00AM - 9:57AM –

Session A33 Drops I: Pinch-off and Coalescence 404 - Jose Gordill, Universidad de Sevilla

8:00AM A33.00001 Scaling transitions during the thinning of viscous dripping droplets[1] , ALFONSO A. CASTREJON-PITA, University of Cambridge, University of Oxford, J. RAFAEL CASTREJON-PITA, University of Cambridge, SUMEET S. THETE, Purdue University, KRISHNARAJ SAMBATH, Chevron Corporation, E. JOHN HINCH, IAN M. HUTCHINGS, JOHN R. LISTER, University of Cambridge, OSMAN A. BASARAN, Purdue University — The dynamics of filament thinning during the formation of viscous Newtonian drops is studied experimentally and numerically. High speed shadowgraph imaging and sub-pixelar image analysis are used to extract the minimum neck diameter in terms of the time τ to breakup. Aqueous solutions of glycerol with viscosities ranging from 20 to 360 mPa s are used as the working fluids. In addition, nozzles of different diameters were used to vary the initial dynamic conditions. High resolution numerical simulations were performed to extract the instantaneous Reynolds number to understand the transition between different scaling regimes (presented in detail in a complementary presentation). Our results seem to suggest that, under some conditions, the transition from Potential Flow (PF) to an inertial-viscous (IV) regime is intertwined by a viscous regime (V).

[1]This work was supported by the EPSRC (Grant EP/G029458/1), the KACST-Cambridge Centre and the Royal Society.

8:13AM A33.00002 Analysis of scaling during pinch-off of Newtonian filaments by numerical simulation , SUMEET THETE, School of Chemical Engineering, Purdue University, USA, KRISHNARAJ SAMBATH, School of Chemical Engineering, Purdue University, USA and Chevron Corporation, USA, OSMAN BASARAN, School of Chemical Engineering, Purdue University, USA, RAFAEL CASTREJÓN-PITA, ALFONSO CASTREJÓN-PITA, IAN HUTCHINGS, Department of Engineering, University of Cambridge, UK, JOHN HINCH, JOHN LISTER, Department of Applied Mathematics and Theoretical Physics, University of Cambridge, UK — Drop formation is ubiquitous in diverse applications, e.g. ink-jet printing and atomization. As a drop is about to form, it is connected by a thinning filament to the rest of the liquid attached to a nozzle. Hence, the physics of filament thinning is key to understanding drop formation. For Newtonian liquids, it is known that the dynamics of filament thinning initially falls in one of two scaling regimes, a potential flow regime if liquid viscosity μ is small or a viscous regime if μ is large. Regardless of μ, the dynamics ultimately transitions to a final asymptotic regime, the inertial−viscous regime, where all forces−viscous, inertial, and surface tension−compete as the filament nears pinch−off. While the global dynamics of drop formation and these scaling regimes are well understood, less well appreciated is what happens during transitions between the initial and final regimes. Here, we investigate the dynamics during these transitions by computation. We also show that computed predictions accord well with experiments detailed in a complementary presentation.

8:26AM A33.00003 Dynamics of Contracting Asymmetric Viscoelastic Filaments , CHRISTOPHER ANTHONY, SUMEET THETE, School of Chemical Engineering, Purdue University, SANTOSH APPATHURAI, Chevron, PRADEEP BHAT, 3M, OSMAN BASARAN, MICHAEL HARRIS, School of Chemical Engineering, Purdue University — In ink-jet printing and atomization, slender filaments are routinely formed. Such filaments either contract to form a single drop or breakup into multiple drops, e.g. by end pinching. Beginning with papers by Schulkes (1996) and Notz & Basaran (2004), past studies have focused exclusively on the contraction dynamics of Newtonian filaments. Also in these studies, initial filament shapes are taken to be long cylinders terminated by two identical spherical caps (symmetric filaments). In emerging applications, e.g. ink-jet printing of complex fluids, the filaments are viscoelastic (VE) fluids. Moreover, older experiments by Notz et al. (2001) and more recent ones by Castrejón-Pita et al. (2012) show that initial filament shapes resemble long, tapered cylinders terminated by hemispherical caps of unequal radii (asymmetric filaments). Therefore, we analyze the contraction dynamics of both asymmetric and symmetric filaments of VE fluids using the Giesekus model. Rather than solving the full set of equations governing the problem, we take advantage of filament slenderness and solve a much simpler set of 1D equations (Eggers, 1997). We then use a finite element method with Streamline Upwind/Petrov Galerkin (SUPG) formulation (Brooks & Hughes, 1982) to solve the reduced equations.

8:39AM A33.00004 Dynamics of contracting surfactant-covered filaments , PRITISH KAMAT, SUMEET THETE, School of Chemical Engineering, Purdue University, QI XU, British Petroleum, OSMAN BASARAN, School of Chemical Engineering, Purdue University — When drops are produced from a nozzle, a thin liquid thread connects the primary drop that is about to form to the rest of the liquid in the nozzle. Often, the thread becomes disconnected from both the primary drop and the remnant liquid mass hanging from the nozzle and thereby gives rise to a free filament. Due to surface tension, the free filament then contracts or recoils. During recoil, the filament can either contract into a single satellite droplet or break up into several small satellites. Such satellite droplets are undesirable in applications where they can, for example, cause misting in a manufacturing environment and mar product quality in ink-jet printing. In many applications, the filaments are coated with a monolayer of surfactant. In this work, we study the dynamics of contraction of slender filaments of a Newtonian fluid that are covered with a monolayer of surfactant when the surrounding fluid is a passive gas. Taking advantage of the fact that the filaments are long and slender, we use a 1D-slender-jet approximation of the governing system of equations consisting of the Navier-Stokes system and the convection-diffusion equation for surfactant transport. We solve the 1D system of equations by a finite element based numerical method.

8:52AM A33.00005 Stretching and Rupture of Suspension Bridges, of the Fluid Variety , KEVIN CONNINGTON, The Levich Institute, The City College of New York, MARK MISKIN, The University of Chicago, TAEHUN LEE, Mechanical Engineering, The City College of New York, MARK SHATTUCK, JEFFREY MORRIS, The Levich Institute, The City College of New York, HEINRICH JAEGER, The University of Chicago — A "suspension bridge" is similar to a liquid bridge but contains solid particles suspended in the liquid. In this work, experiments and numerical simulations are performed to examine the dynamics of the stretching of a suspension bridge, and the eventual rupture. The experiments are performed using a suspension density matched with the surrounding immiscible liquid to minimize gravitational effects; the simulations are performed using a multi-component lattice-Boltzmann(LB) method coupled with an established method for LB simulation of suspended solids. The focus is on particle rearrangements and rupture dynamics, as well as the force required to stretch the bridge, with comparisons made between the case of a suspension bridge and simple liquid bridge. It is found that even under dilute particle loading, the rupture dynamics are significantly altered by the influence of particles. Under concentrated conditions, the rearrangements of the particles are associated with significant distortion of the interface, and a simpler simulation tool which balances particle interactions with the capillary forces from the boundary appears to capture salient features of the dynamics. The ultimate rupture dynamics are compared to the pinch-off behavior in drop formation from suspensions.

9:05AM A33.00006 Inducing coalescence by a superposition of two Rayleigh-Plateau instabilities: Theoretical analysis , THEO DRIESSEN, PASCAL SLEUTEL, FRITS DIJKSMAN, University of Twente, ROGER JEURISSEN, Eindhoven university of technology, DETLEF LOHSE, University of Twente — We demonstrate a novel method of producing a stream of widely spaced high-velocity droplets by imposing a combination of two unstable modes on a liquid jet. The wavelengths of the two modes are chosen close to the wavelength of the most unstable mode. After the initial breakup of the jet into small droplets, these droplets coalesce to produce a stream of larger droplets spaced at a much larger distance than the wavelength of the most unstable mode of the jet. We analytically derive sets of perturbations that robustly induce this process, and we investigate the influence of the nonlinear interactions in the Rayleigh-Plateau instabilities on the coalescence process. Experiments and numerical simulations demonstrate that the jet breakup and the subsequent droplet merging are governed completely by the selected modes.

9:18AM A33.00007 Inducing coalescence by a superposition of two Rayleigh-Plateau instabilities: Experimental implementation , PASCAL SLEUTEL, THEO DRIESSEN, University of Twente, ROGER JEURISSEN, ACFD Consultancy, FRITS DIJKSMAN, DETLEF LOHSE, University of Twente — In this work we present an experimental method to efficiently breakup and coalesce multiple droplets from a jet by a superposition of two Rayleigh-Plateau perturbations. A continuous liquid jet is ejected from a glass capillary which has a piezo electric actuator attached to it. The periodical pressure perturbations applied by the piezo induce two growing modes on the jet. By choosing the perturbation wavenumbers close to wavelength of the most unstable mode, fast coalescence and a stable stream of droplets are obtained. By tuning the phase between the two perturbations we control the coalescence time and the satellite droplet formation. When the coalescence process is finished, the final droplet size is set by the low frequency beating wavelength. This means that stable streams of mono-disperse droplets can be generated at inter-droplet distances and droplet velocities very different from a single Rayleigh-Plateau instability. Our experimental results are compared with numerical results and there is agreement in great detail.

9:31AM A33.00008 Multiscale computations of thin films between colliding drops[1] , BAHMAN ABOULHASANZADEH, SADEGH DABIRI, GRETAR TRYGGVASON, University of Notre Dame — In multiphase flows thin films frequently appear between fluid blobs colliding with each other. These films can become very thin and be difficult to resolve accurately in numerical simulations, particularly in DNS of many co-flowing drops, requiring very fine resolution and resulting in excessive computational cost due to very fine uniform grids or time consuming adaptive mesh refinement. Here, we describe an algorithm for detecting thin films using a front tracking method. We also propose a subscale model to describe the physics and the evolution of a thin film between two drops. Comparison between results for a fully resolved film on a fine grid and simulations using a much coarser grid plus the model for the description of the film, shows good agreement.

[1]This study was funded in part by NSF Grant CTS-0522581.

9:44AM A33.00009 Transition from partial to complete coalescence , BAHNI RAY, TAEHUN LEE, Department of Mechanical Engineering, City College of City University of New York, USA — The lattice Boltzmann equation (LBE) method is used to simulate satellite drop formed during coalescence of unequal size drops first shown experimentally by Zhang, Li and Thoroddsen [Phys. Rev. Lett. **102**, 104502 (2009)]. Partial coalescence is commonly observed for drop impact on flat surface for a particular range of initial drop diameter. Important criterion for partial coalescence is the increasing horizontal momentum of the drop relative to the vertical momentum. The experimental observation of similar phenomena with two unequal size drops emphasize on the fact that the curvature of the surface has an important contribution as well. Simulations are performed to show that the drop curvature and drop liquid drainage time effects the satellite drop formation. Furthermore the study is extended to drops with surrounding liquid medium and compared to drop coalescence on flat surface.

Sunday, November 24, 2013 8:00AM - 9:57AM –

Session A34 Drops II: Drop Impact on Liquid Surfaces 405 - Yongsheng Lian, Louisville University

8:00AM A34.00001 The effects of droplet characteristics on the surface features in a rain field[1] , R. LIU, H. BROWN, X. LIU, J.H. DUNCAN, University of Maryland — The characteristics of the shape of a water surface in response to the impact of simulated raindrops are studied experimentally in a 1.22-m-by-1.22-m water pool with a water depth of 0.3 m. A rain generator consisting of an open-surface water tank with an array of 22-gauge hypodermic needles (typical needle-to-needle spacing of about $L_0 = 3.5$ cm) attached to holes in the tank bottom is mounted 2 m above the water pool. The tank is connected to a 2D translation stage to provide a small-radius ($< L_0$) horizontal circular motion to the needles, thus avoiding repeated drop impacts at the same location under each needle. The droplet diameters (d) and number of drops per unit time per needle (n) are varied by changing the length of the needles while maintaining the same volumetric flow rate ($n\pi d^3/6$) through control of the water depth in the generator tank. The water surface features, including the crown, stalk and ring waves, due to the impacts of the drops are measured with a cinematic laser-induced-fluorescence (LIF) technique. The dependence of these features on the rain characteristics are discussed.

[1]The support of the National Science Foundation, Division of Ocean Sciences, and the assistance of Mr. Larry Gong are gratefully acknowledged.

8:13AM A34.00002 Numerical investigation of air film breakup and micro-bubble formation in liquid-liquid impact events[1] , SEYEDSHAHABADDIN MIRJALILI, ALI MANI, Stanford University — Experimental evidence shows that micro-bubbles can be generated when a droplet of the size of a few millimeters impacts a layer of the same liquid with a velocity of a few meters per second. This phenomenon, also known as Mesler entrainment is cumbersome to numerically simulate due to the small time and length scales involved. In order to gain a better understanding of the relevant scales, parameters, and regions, 2-D boundary element simulations inspired by M. Mani, Mandre, Brenner (JFM, vol. 647, p. 143, 2010) were performed. By developing treatments for topological changes, these simulations are extended to after impact events and finally depict the formation of micro-bubbles of sizes similar to entrapped bubbles in Mesler entrainment. Compressibility effects on final bubble size are discussed, and the requirements for a resolved CFD calculation are obtained. Thereafter, a 2-D two-phase flow calculation using a diffuse interface model is undertaken and based on grid-converged results, the statistics of the bubbles are examined and compared with available experimental data.

[1]Supported by the Office of Naval Research.

8:26AM A34.00003 Dynamics of an air film entrapped by drop impact on liquid surface , JI SAN LEE, Postech, BYUNG MOOK WEON, Sungkyunkwan University, SU JI PARK, JI WON JUNG, JI TAE KIM, JAEYEON PYO, POSTECH, KAMEL FEZZAA, Argonne National Laboratory, JUNG HO JE, POSTECH — When a liquid drop impacts a liquid substrate, air is entrapped underneath, finally evolving into spherical bubbles. This phenomenon occurs robustly and plays an important role in various natural phenomena and industrial applications. In this study, we investigated the evolution of an entrapped air film during drop impact using ultrafast X-ray phase-contrast imaging. The evolution exhibits very complex and different behaviors depending on fluid properties. We reveal that the retraction dynamics of the air film crosses from a capillary-inertial regime to a capillary-viscous regime by the increase of Ohnesorge number (Oh). At Oh <0.03, a daughter droplet is generated inside the air film, due to the convergence of the capillary waves generated during the retraction. We also find that the evolved bubble is broken up into two at low Oh, driven by its vertical stretching and capillary instability. We finally demonstrate a phase diagram for the formation of daughter droplet and bubble breakup in terms of Oh and Weber number (We).

8:39AM A34.00004 Surfactant effects on cumulative drop size distributions produced by air bubbles bursting on a non-quiescent free surface[1] , K. PARMAR, X. LIU, J.H. DUNCAN, University of Maryland — The generation of droplets when air bubbles travel upwards from within a liquid and burst at a free surface is studied experimentally. The bubbles are generated in a glass water tank that is 0.91 m long and 0.46 m wide with a water depth of 0.5 m. The tank is equipped with an acrylic box at its bottom that creates the bubble field using filtered air injected through an array of 180 hypodermic needles (0.33 mm ID). Two different surface conditions are created by using clean water and a 0.4% aqueous solution of Triton X-100 surfactant. Measurements of the bubble diameters as they approach the free surface are obtained with diffuse light shadowgraph images. The range of bubble diameters studied is 2.885 mm to 3.301 mm for clean water and 2.369 mm to 3.014 mm for the surfactant solution. A laser-light high-speed cinematic shadowgraph system is employed to record and measure the diameters and motions of the droplets at the free surface. This system can measure droplets with diameters $\leq$ 50 μm. The results show a clear distinction between the droplet distributions obtained in clean water and the surfactant solution. A bimodal droplet distribution is observed for clean water with at least two dominating peaks. For the surfactant solution, a single distribution peak is seen.

[1]This work is supported by the National Science Foundation, Division of Ocean Sciences.

8:52AM A34.00005 Sparkling Droplets: Aerosol Dispersion Resulting from Drop Impingement on Porous Surfaces , YOUNG SOO JOUNG, CULLEN BUIE, Massachusetts Institute of Technology — We have investigated aerosol generation from droplets hitting wettable porous surfaces. Aerosols have been widely investigated due to their significant impact on the environment. To date, bubbles breaking at air/water interfaces have been considered the chief mechanism of aerosol dispersion. Here, we demonstrate that droplets can release aerosols when they impact porous surfaces. At the moment of impact, tiny bubbles are formed inside the droplet, fed by air escaping the porous media. The tiny bubbles break when they meet the droplet/air interface, releasing tiny water-jets, the sizes of which are in the range of tens-of-micrometers. Interestingly, within a specific range of impact velocity and surface properties, we observed frenzied ejection of tiny water-jets producing aerosol clouds above the surface. With knowledge of the surface properties and the impact conditions we can predict when the frenzied aerosol generation will occur. This study will produce novel experimental methodologies for further investigation of the environmental impact of aerosols.

9:05AM A34.00006 Rain Drops and Oil Slicks: Impact of Water Droplets on a Surface Oil Layer[1] , DAVID MURPHY, DAVID MORRA, JOSEPH KATZ, Johns Hopkins University — Petroleum spills in aquatic environments form oil slicks on the water surface. These slicks, the thickness of which ranges from microns to several millimeters, negatively impact the natural environment and economic resources. While dispersion of these slicks as small droplets by breaking waves has long been investigated, the dispersive power of another environmental flow, rainfall, has not been considered. The impact of a water drop on a floating layer of immiscible fluid introduces a challenging flow physics problem. Our experimental observations examine processes occurring when falling water droplets impact on floating layers of sweet petroleum crude oil of various thicknesses and dispersant concentrations. The latter alter the surface tension by orders of magnitude. Impact events recorded at high-speed, using UV light to cause oil fluorescence, show the expected formation of modified multiphase Worthington jets, air cavities, as well as breakup of the slicks into clouds of oil droplets and oil-coated bubbles. The latter rise back to the surface and pop. Results include droplet size and spatial distributions as a function of rainfall momentum, oil properties, and processes involved.

[1]Sponsored by Gulf of Mexico Research Initiative (GoMRI)

9:18AM A34.00007 Droplet splashing on a wet moving wall , JING LOU, MING CHENG, Institute of High Performance Computing, Agency for Science, Technology and Research (A*STAR) — The splash of a liquid drop onto a moving solid wall covered with a thin film of the same liquid is simulated numerically for an impact Reynolds number of 1000 and a Weber number of 400. A two-phase flow lattice Boltzmann model is employed for the simulations. The characteristics of drop splashing on the wall are investigated by systematically varying the impact angle and wall speed. The angle of incidence θ ranges from 0 to 60 degrees, while the wall speed to the initial drop velocity ratio Uw varies from 0 to 3. In case of Uw equals to 0 and θlarge than 0 degree, the splashing is asymmetric. The increase of the angle leads to the transition from splash to partial splash. The present results indicate the existence of a critical θc value of about 55 degrees, which is determined by the liquid film thickness, Reynolds and Weber numbers.

9:31AM A34.00008 Droplets motion by Dissipative particle dynamics on inclined flat plate with oil film covered , CHUANJIN LAN, YANBAO MA, University of California, Merced — Understanding the incipient motion of a droplet adhered to a surface has wide applications, from self-cleaning of superhydrophobic surfaces to electromagnetic drop actuation inmicrofluidic systems. In this paper the dissipative particle dynamics (DPD) is introduced to study the droplet motion at the meso-scale along the inclined flat plate covered with oil film at different tilt angles. The flow motion can be totally different with and without the existence of the oil film. The effect from tilt angle and oil film thickness is studied on the motion of the droplet.

9:44AM A34.00009 Can consecutive droplet deposition yield uniform liquid films?[1] , ANNE JUEL, ALICE THOMPSON, CARL TIPTON, ANDREW HAZEL, University of Manchester, MARK DOWLING, Cambridge Display Technology — Ink-jet printing is being used to manufacture highly customizable electronic components through the successional deposition of overlapping droplets of conductive ink onto a substrate. For vanishing contact angles, the fluid drops merge together on the substrate to form a wide, uniform thread. However, in practical operating regimes involving moderate contact angles, the fluid is redistributed to form non-uniform patterns, characterized by the development of an initial wide bulge (larger than any individual drop). This "head" is sometimes followed by a sequence of secondary bulges connected by narrow rivulets. We conduct new experiments with high-speed imaging to examine the morphological development of the thread resulting from the deposition of a moderate number of successive micro-scale drops (< 80) that may form up to two secondary bulges. We demonstrate that the spatial structure of the ink formation resulting from this complex deposition process can be predicted quantitatively with a simple physical model that incorporates contact angle hysteresis. The head formation occurs robustly due to an inviscid mechanism, while the secondary bulge formation arises from viscous effects.

[1]EPSRC

Sunday, November 24, 2013 8:00AM - 9:44AM –

Session A35 Suspensions I: Structure and Phase Transitions 406 -

8:00AM A35.00001 An immersed boundary method for investigating the rheology of heavy crude oil[1] , MOHSEN DAGHOOGHI, IMAN BORAZJANI, University at Buffalo — Heavy crude oil is a colloidal suspension of aggregates of many compounds (resins and asphaltenes), which give rise to very unusual rheological properties such as non-Newtonian behavior and very high viscosity. To optimize processes to lower viscosity and improve flow properties of heavy crude oil, a better understanding of the relationship between micro-structure and macro rheological behavior is required. To date, this relationship has been investigated mostly using theoretical models with many simplifying assumptions on the shape, size, and concentration of aggregates that limit their validity and use. We extend our immersed boundary method to simulate arbitrarily shaped suspensions of particles and calculate the viscosity of the suspension. We validate our method against the classical analytical and experimental results for the low Reynolds-Stokes problem of particle suspensions based on the work of Einstein, Batchelor, and others. We apply our method to simulate colloidal suspensions of asphaltenes with their experimentally observed micro-structure. We investigate the effects of asphaltene aggregates' concentration, size, shape, and polydispersity (different aggregate sizes) on the viscosity of the heavy crude oil.

[1]This work was supported by American Chemical Society.

8:13AM A35.00002 Acoustic Properties of Dilute Microstructured Suspensions: Theory and Experiment , WUHAN YUAN, LIPING LIU, JERRY SHAN, Rutgers University — It is known that the orientation of ellipsoidal ferromagnetic particles in suspensions can be readily manipulated by external magnetic fields. This variable suspension microstructure can lead to a novel acoustic medium with tunable sound speed and attenuation. In pursuit of a better understanding of the acoustic properties of such microstructured suspensions, we develop a theoretical model for the effective viscosity of suspensions of oriented ellipsoidal particles. We then use this model to predict the acoustic properties of the suspensions, and how they depend upon sound frequency and particle volume fraction, aspect ratio, and orientation. On the other hand, we also conduct a series of experiments measuring sound speed and attenuation in suspensions of nickel microflakes with and without microstructure induced by external magnetic fields. The experimental and theoretical results are presented and compared.

8:26AM A35.00003 Derivation of the rheological properties of a dilute suspension of spheres in a dilute polymer solution using the method of ensemble averaged equations , DONALD KOCH, ERIC LEE, Cornell University, IBRAHIM MUSTAFA, Yanbu Industrial College — Einstein derived the first effects of spherical particles on the rheology of a Newtonian fluid in the limit of small particle concentration. In the past, the only comparable analysis for a non-Newtonian fluid considered a second order fluid constitutive equation valid for very low Deborah number (defined as the shear rate times the polymer relaxation time). In this paper, the ensemble average stress of a dilute suspension of spheres in a dilute polymer solution governed by the Oldroyd B rheological equation is derived for Deborah numbers up to 4. The extra stress in the suspension arises from three physical processes: the stretching of the polymers due to the disturbance flow of the particles, the increased particle stresslet due to the polymer stress, and the modification of the Newtonian stress due to the flow perturbation caused by the polymers. We make use of an asymptotic analysis for small polymer concentration and the generalized reciprocal theorem to derive the third contribution. While a particle-free Oldroyd B fluid has no shear thinning or thickening, the particle suspension exhibits shear thickening and a nonlinear increase in the second normal stress difference with shear rate.

8:39AM A35.00004 Rheology and fluid mechanics of a hyper-concentrated biomass suspension , LORENZO BOTTO, School of Engineering and Materials Science, Queen Mary, University of London, XIAO XU, Department of Chemical Engineering, Imperial College, London — The production of bioethanol from biomass material originating from energy crops requires mixing of highly concentrated suspensions, which are composed of millimetre-sized lignocellulosic fibers. In these applications, the solid concentration is typically extremely high. Owing to the large particle porosity, for a solid mass concentration slightly larger than 10%, the dispersed solid phase can fill the available space almost completely. To extract input parameters for simulations, we have carried out rheological measurements of a lignocellulosic suspension of Miscanthus, a fast-growing plant, for particle concentrations close to maximum random packing. We find that in this regime the rheometric curves exhibit features similar to those observed in model "gravitational suspensions," including viscoplastic behaviour, strong shear-banding, non-continuum effects, and a marked influence of the particle weight. In the talk, these aspects will be examined in some detail, and differences between Miscanthus and corn stover, currently the most industrially relevant biomass substrate, briefly discussed. We will also comment on values of the Reynolds and Oldroyd numbers found in biofuel applications, and the flow patterns expected for these parameter values.

8:52AM A35.00005 Microstructure and rheology of colloidal suspension in simple shear and dynamic oscillatory flows: theory and simulation , EHSSAN NAZOCKDAST, Courant Institute, New York University, STEPHANIE MARENNE, JEFFREY MORRIS, Levich Institute and Department of Chemical Engineering at CUNY — A Smoluchowski theory is developed for analytical prediction of structure and rheology of concentrated colloidal dispersions. Pair distribution function, $g(\mathbf{r})$, is computed as a solution to pair Smoluchowski convection-diffusion equation at a given volume fraction, ϕ, and *Pe* which is the ratio of hydrodynamic to Brownian forces. Pair distribution function is then used to evaluate rheology. Many body interactions are modeled self-consistently through third particle integrals which allows for predictions of structure and rheology at $\phi < 0.50$ and the entire range of *Pe*. The predictions of structure and rheology of steady state simple shear flow are in good agreement with Accelerated Stokesian Dynamics simulations (ASD). In this work the theory is extended to study time-dependent pair structure, $g(\mathbf{r}\ t)$, and rheology of colloidal dispersions, considering both startup of steady shear and oscillatory shear flows. The predictions of startup flow are compared against ASD simulations over a wide range of *Pe* and ϕ. The predictions of stress and microstructure in oscillatory shear flow are also presented over a wide range of strain amplitudes and frequencies.

9:05AM A35.00006 Reversible plastic regime in a 2D jammed material[1] , NATHAN KEIM, PAULO ARRATIA, University of Pennsylvania — At the microscopic level, flow of a jammed, disordered material consists of a series of particle rearrangements that cannot be reversed. The same material under infinitesimal deformation is free of rearrangements, perfectly reversible, and dominated by elastic stress. Yet several recent studies have found an intermediate regime with observable plastic activity microscopically, but not globally: there is no net change to the material upon reversing the deformation. We report on the occurrence and structure of these reversible plastic events in experiments with an interfacial material, which do not give rise to global irreversibility. Reversible plasticity couples to the bulk shear stress and so contributes to bulk dissipation and viscoplasticity — but this is the case for only a limited range of strain amplitudes below and above the yielding transition.

[1]This work was supported by the Penn NSF MRSEC (DMR-1120901).

9:18AM A35.00007 Analogy between strain-stiffening and jamming in dense flows , GUSTAVO DÜRING, Facultad de Física, Pontificia Universidad Católica de Chile, EDAN LERNER, MATTHIEU WYART, CSMR, New York University, New York, USA — Dense granular and suspension flows display peculiar properties near the jamming threshold: the rheology is singular, and a diverging length scale can be identified from the velocity correlation of the particles, or from non-local effects affecting flow. In elastic networks a rigidity transition occurs when the coordination z is increased toward some threshold z_c, but can also take place if a large strain is imposed as remarked early on by Maxwell. The latter phenomenon has been proposed to cause the ubiquitous stress-stiffening observed in gels of biopolymers. In my talk I will present the critical properties of a network immersed in a fluid approaching such a strain-induced rigidity transition. Then I will argue that this transition is at play in dense suspension flows, where it corresponds microscopically to the buckling of force chains. Our predictions include the existence of a vanishing strain $\gamma \sim 1/p$ in flow near jamming, where p is the dimensionless particle pressure, and unravels the existence of two length scales affecting flow.

9:31AM A35.00008 Age coarsening of colloidal gels: a micro-mechanical study , ROSEANNA ZIA, Cornell University, BENJAMIN LANDRUM, WILLIAM RUSSEL, Princeton University — We study the evolving structure and time-dependent rheological properties of an aging colloidal gel, with a focus on understanding the non-equilibrium forces that drive late-age coarsening. The gel is formed from a dispersion of Brownian hard spheres that interact via a hard-sphere repulsion and short-range attraction. The O(kT) strength of attractions lead to an arrested phase separation, and the resulting structure is a bi-continuous, space-spanning network that exhibits elastic and viscous behaviors: the gel may sustain its weight under gravity, or flow under shear. With weak attractions the bonds are reversible, giving rise to a continuous breakage / formation process as the gel ages. This balance favors coarsening over time, accompanied by an increase in feature size and elastic strength. We show here that anisotropic surface migration leads to heterogeneous coarsening, and that this migration is driven by gradients in particle-phase stress.

Sunday, November 24, 2013 8:00AM - 9:57AM –

Session A36 Geophysical: Oceanographic I 407 - Peter Diamessis, Cornell University

8:00AM A36.00001 Modifications to Symmetric and Baroclinic Instabilities in the Presence of Surface Gravity Waves , SEAN HANEY, Department of Atmospheric and Oceanic Sciences, University of Colorado, Boulder, CO, BAYLOR FOX-KEMPER, Department of Geological Sciences, Brown University, Providence, RI, KEITH JULIEN, Department of Applied Mathema-cs, University of Colorado, Boulder, CO — The depth of the ocean mixed layer is determined by several processes that mix and restratify the ocean. The classical Eady problem (linear stability of flow in thermal wind balance) describes the growth of baroclinic instabilities which are important for restratification in the mixed layer. The Eady problem has been extended by Stone to include non-hydrostatic effects and a range of Richardson numbers appropriate for the ocean mixed layer. Here, the problem is extended further to include the Stokes drift (a drift current that is the time averaged effect of the surface gravity waves). A new, wave-induced, instability is introduced that coexists with the symmetric and baroclinic instabilities. In addition, both the symmetric and baroclinic instabilities are modified by the presence of Stokes drift. While the baroclinic mode becomes a hybrid baroclinic/wave-induced mode at modest wave forcing, the symmetric mode is only slightly modified. This transition to a hybrid mode is marked by a change in the energy source and the vertical structure. Dominance by each of the three modes may occur in a realistic parameter regime for the ocean mixed layer, and therefore wave forcing cannot be neglected when considering the stability of the mixed layer.

8:13AM A36.00002 Sea-surface manifestation of a submerged stratified turbulent wake via wake-emitted internal gravity waves[1] , QI ZHOU, PETER DIAMESSIS, Cornell University — A submerged turbulent wake in the stratified ocean may become visible at the sea surface due to the internal gravity waves (IGWs) which are emitted by the wake and propagate towards the surface. In a linearly stratified Boussinesq fluid, we examine such a wake and wake-emitted IGWs at wake Reynolds number $Re \in [5 \times 10^3, 10^5]$ and Froude number $Fr \in [4, 16, 64]$ using three-dimensional implicit Large Eddy Simulations. A spectral multidomain penalty scheme in the vertical enables finer resolution of both the IGW-emitting wake and the subsurface region where the IGWs interact with a free-slip sea surface. At various wake parameters, including Re, Fr and the evolution stage of a wake, we report the length- and time-scales of reflecting IGWs at the surface, statistics of magnitudes and orientations of IGW-induced surface strains, and mean momentum fluxes due to IGWs. A case study concerning the visibility of the surfacing IGWs from remote sensors is performed by considering possible local enrichment of surfactant due to the surface IGW currents/strains.

[1]Work funded by ONR grant N00014-08-1-0235 (Dr. Ron Joslin)

8:26AM A36.00003 The surface generation and downward propagation of internal waves in nonlinear stratifications , SASAN GHAEMSAIDI, THOMAS PEACOCK, MIT, THIERRY DAUXOIS, SYLVAIN JOUBAUD, PHILIPPE ODIER, ENS Lyon — An important topic in physical oceanography is the generation of internal waves by surface forcing, and the subsequent propagation of these waves into the deep ocean, often through complex density stratifications. This scenario is of particular interest in the Arctic Ocean, where increased summer ice loss is leading to enhanced internal wave activity, which in turn impacts circulation via wave breaking and mixing. We present the results of a combined theoretical and laboratory experimental study of this scenario, seeking to identify key parameters that significantly influence the amplitude of the wave field transmitted to the deep ocean.

8:39AM A36.00004 Characterization of the Vertical Energy Distribution of the Internal Wave Field in the Upper Ocean , JEREMY BRUCH[1], Johns Hopkins University — A method to simulate internal waves in the upper ocean is proposed by defining the vertical energy distribution as a function of mode number with the associated vertical structure functions as an appropriate set of orthogonal basis functions. An internal wave simulation is shown for a case with a stylized BV peak profile, using the Garrett and Munk internal wave model (GM) as the input energy distribution. The resulting simulated spectra are shown to be self-consistent with the proposed definition of the vertical energy distribution. Application of the GM model requires many assumptions, including the requirement that the internal waves are modeled strictly in deep water where there is little variation in the stratification. Given the typical non-uniformity of the stratification profile in the upper ocean, it may be of interest to relax this restriction of the GM model but the obvious non-stationary properties near the thermocline are incompatible with the calculation of the vertical spectrum of the internal wave field. The method described in this presentation suggests a means to reconcile this incompatibility.

[1]Membership pending

8:52AM A36.00005 Internal Solitary Wave Tunnelling , BRUCE SUTHERLAND, SCOTT KEATING, University of Alberta, ISHITA SHRIVISTAVA[1], IIT Mumbai — In a two-layer fluid, solitary waves of depression (elevation) propagate in a shallow upper (lower) layer. The transition from depressed to elevated is known to occur as a solitary wave of depression passes over a bottom slope. If impacting a coastline the shoaling waves deposit some energy and partially reflect. Here we consider what happens if a solitary wave passes over a sill or the shoulder of an island. Specifically, through lock-release laboratory experiments, we examine the evolution of a solitary wave of depression incident upon a submerged thin vertical barrier and triangular submarine topography. From the measured interface displacement, we determine the available potential energy associated with the wave. The method of Hilbert transforms is used to subdivide the displacement signal into rightward- and leftward-propagating disturbances, from which we measure the available potential energy of the transmitted and reflected waves. These are used to measure the relative transmission, reflection and deposition of energy in terms of the barrier height and slope, the relative depths of the ambient fluid and the amplitude of the incident wave. Implications for internal wave scattering around Dongsha Atoll in the South China Sea are discussed.

[1]Research performed while visiting the University of Alberta under the UARE program

9:05AM A36.00006 Mass transport by large and very-large amplitude mode-2 internal solitary waves: experimental observations , KARA SHIPLEY, ALAN BRANDT, JHU/APL — The present experiments provide the first quantitative measurements of the mass transport by mode-2 internal solitary waves (ISW) propagating on a thin pycnocline. The ISW were generated by the release of fluid from an initially mixed volume. It was found that the amplitude and amount of mass transported by the leading and second, following ISW was proportional to the level of forcing and was attenuated at an approximately uniform rate as the ISW propagated downstream. At the highest level of ISW forcing, over 40% of the mixed fluid was transported within the leading ISW. Excellent agreement was found with the numerical simulations of Salloum et al. (2012) that were designed to replicate the present experimental configuration. In addition, a new ISW regime was identified, termed very large-amplitude ISW, where the ISW bulge wavelength and extent of mass transported increased with amplitude at a rate greater than the lesser amplitude ISW. In recent years the frequent occurrence of large amplitude ISW in the coastal ocean has been observed. The present experiments and the associated numerical simulations can provide insight into the effects of ISW transport on coastal mixing and biological material distribution.

9:18AM A36.00007 Simultaneous experimental measurements of velocity and density in solitary internal waves with trapped cores , P. LUZZATTO-FEGIZ, University of Cambridge, K. HELFRICH, Woods Hole Oceanographic Institution — Long internal waves with trapped cores are relatively common in the ocean and atmosphere (e.g. Lien et al. 2012). It has been proposed that such waves may be important for transporting mass, energy, and biological matter across the continental shelf (Shroyer et al. 2010, Scotti & Pineda 2004). However, several fundamental wave properties, including mass and energy transport, as well as core circulation and density structure, remain to be quantified experimentally. A key prerequisite, for such measurements, involves simultaneously accessing the velocity and density fields with sufficient resolution. We employ a setup comprising a thin linearly stratified region overlaying a deep, uniform-density layer, and perform experiments with and without a no-slip lid at the surface. The waves are produced by a lock-release mechanism. We develop a technique for high-resolution, simultaneous measurements of velocity and density in stratified flows, using pulsed-laser, co-planar PIV and LIF. We are thereby able to extract properties including phase velocity, kinetic and potential energies, minimum Richardson number, as well as core size, circulation and density. To examine larger waves, we complement these results with numerical simulations, which are in good agreement with our experiments.

9:31AM A36.00008 Effect of External Turbulence on the Evolution of a Towed Wake in a Stratified Environment , ANIKESH PAL, SUTANU SARKAR, University of California, San Diego — Direct numerical simulation (DNS) is used to study the effect of external turbulence on the evolution of a towed turbulent wake in a stratified fluid. The simulations are carried out at a Reynolds number of 10,000, Froude number of 3 and Prandtl number of 1. The external turbulence is generated from a triply periodic rectangular domain in an auxiliary simulation performed to obtain turbulence with desired $u^{'}_{ext}/U_0$, where $u^{'}_{ext}$ is the root mean square velocity of the external turbulence and U_0 is the maximum defect velocity of the pure towed wake. This field of external turbulence is added to the initial field of the towed turbulent wake. Simulations are performed for $u^{'}_{ext}/U_0 = 0.10, 0.20$ and 0.30. The kinetic energy of the towed wake decays faster with progressively increasing values of $u^{'}_{ext}/U_0$. This effect of external turbulence is found to be stronger in stratified flow relative to the neutral case. Although the horizontal spread of the stratified wake is enhanced owing to external turbulence there is little effect on the vertical spread.

9:44AM A36.00009 Drag Coefficients of Drifting Waterbirds[1] , KARL VON ELLENRIEDER, Florida Atlantic University, KEVIN KENOW, U.S. Geological Survey, HUAJIN (ARIEL) QU, TSUNG-CHOW (JOE) SU, Florida Atlantic University — A series of towing tank experiments has been performed to support the development of a probabilistic source tracking model that can be used to estimate the origin of waterbird die-offs. While monitoring the appearance of waterbird carcasses on beaches provides the primary means of assessing the magnitude, as well as the spatial and temporal patterns of die-offs, interpreting the actual site of exposure to toxins is hampered by a lack of information on the drift patterns of carcasses and the confounding influences of wind/current. In this work, a series of experimental measurements were conducted on Common Loon and Lesser Scaup carcasses to obtain steady drag coefficients of representative waterbird species. The tests were designed to capture the drag coefficients associated with current speeds of between 0.2 and 0.8 meters per second and wind speeds of up to 10 meters per second at different levels of carcass submergence. Using the submerged frontal area of an ellipse, together with the frontal area of any submerged portions of the head and neck gives good similarity across the ranges of speeds and submergence levels tested. An example approach to determining waterbird drift velocity and direction from knowledge of the drag coefficients, wind and current is provided.

[1]This effort was supported by the US Geological Survey

Sunday, November 24, 2013 10:25AM - 12:20PM –

Session B27 Awards Presentation, followed by the Otto Laporte Lecture and Corrsin Award Lecture Spirit of Pittsburgh - Neelesh Patankar Northwestern University, Gretar Tryggvason, University of Notre Dame

10:25AM B27.00001 Introduction –

10:50AM B27.00002 Fluid Dynamics Prize Talk: The Reactive Flow of Ideas[1] , ELAINE ORAN, University of Maryland and Emeritus, US Naval Research Laboratory — This presentation describes the evolution of our understanding of several key ideas in reactive flow from *Ignorance* to *Discovery* to *Application* and then again, to *Ignorance*. These key ideas describe the interactions of shock waves and turbulence with reaction fronts, and explain mechanisms for dynamic changes in the fundamental nature of the flow. They explain how flames undergo transitions from small ignition sources to turbulent flames to detonations, and how these energetic reactions waves may decay and die. Applications of the key ideas have been used to explain phenomena ranging from supernova explosions to catastrophic accidents in chemical plants. They have also helped to develop strategies for ensuring safety when we deal with energetic materials, and to create engines for high-speed flight. Now, however, we are at a turning point: By combining experimental observations with the most recent results of theory, advances in computational algorithms, and the ability to do large-scale numerical simulations, discrepancies arise that challenge well-established equations and approaches, both fluid and chemical. And so we must now ask: *What is the origin of these discrepancies? What do we do next?*

[1]With many thanks to friends and colleagues, and to NRL, ONR, AFOSR, and NASA for their support.

11:35AM B27.00003 Stanley Corrsin Award Talk: Fluid Mechanics of Fungi and Slime , MICHAEL BRENNER, Harvard University — There are interesting fluid mechanics problems everywhere, even in the most lowly and hidden corners of forest floors. Here I discuss some questions we have been working on in recent years involving fungi and slime. A critical issue for the ecology of fungi and slime is nutrient availability: nutrient sources are highly heterogeneous, and strategies are necessary to find food when it runs out. In the fungal phylum *Ascomycota*, spore dispersal is the primary mechanism for finding new food sources. The defining feature of this phylum is the ascus, a fluid filled sac from which spores are ejected, through a build up in osmotic pressure. We outline the (largely fluid mechanical) design constraints on this ejection strategy, and demonstrate how it provides strong constraints for the diverse morphologies of spores and asci found in nature. The core of the argument revisits a classical problem in elastohydrodynamic lubrication from a different perspective. A completely different strategy for finding new nutrient is found by slime molds and fungi that stretch out – as a single organism– over enormous areas (up to hectares) over forest floors. As a model problem we study the slime mold *Physarum polycephalum*,which forages with a large network of connected tubes on the forest floors. Localized regions in the network find nutrient sources and then pump the nutrients throughout the entire organism. We discuss fluid mechanical mechanisms for coordinating this transport, which generalize peristalsis to pumping in a heterogeneous network. We give a preliminary discussion to how physarum can detect a nutrient source and pump the nutrient throughout the organism.

Sunday, November 24, 2013 1:35PM - 2:10PM –

Session C27 Invited Session: Roughness Effects on Wall-Bounded Turbulent Flows Spirit of Pittsburgh Ballroom A - James M. Wallace, University of Maryland

1:35PM C27.00001 Roughness Effects on Wall-Bounded Turbulent Flows[1] , KAREN FLACK, United States Naval Academy — The importance of surface roughness is well known for wall-bounded flows. Roughness typically increases drag in turbulent boundary layers due to pressure forces on the roughness elements. While rough-wall flows are ubiquitous in engineering practice, the issues of modeling the roughness in computations and accurately predicting the increase in frictional drag remain elusive goals. In this talk, the effect of roughness on the mean flow, turbulence statistics, and turbulence structure will be discussed. In particular, rough-wall flows will be examined in light of Townsend's Reynolds number similarity hypothesis, which states that the turbulent motions in the outer layer are independent of surface roughness when the Reynolds number is sufficiently high. Additionally, the presentation will include recent work on the estimation of frictional drag due to surface roughness. Detailed experiments have been performed in the transitionally rough and fully rough regimes. This research is part of an effort to determine the relevant predictive scales based solely on the roughness topography.

[1]Work supported by the Office of Naval Research

Sunday, November 24, 2013 1:35PM - 2:10PM –

Session C28 Invited Session: Microfluidic Flows of Complex Suspensions: from Flexible Polymers to Swimming Bacteria Spirit of Pittsburgh Ballroom B/C - Howard A. Stone, Princeton University

1:35PM C28.00001 Microfluidic flows of complex suspensions: from flexible polymers to swimming bacteria , ANKE LINDNER, PMMH-ESPCI — The flow of complex suspensions is ubiquitous in nature and industrial applications. Their non-Newtonian character is due to flow-induced orientation, rearrangement, or deformation of microscopic objects suspended in simple fluids. Linking the microstructure on the particle level to the macroscopic response under flow is one of the fundamental scientific challenges of soft matter physics. Here, we present two examples of flows of complex suspensions in chosen microfluidic geometries which allow this link to be established. First, we use a solution of flexible polymers, where normal stresses are known to arise when the polymers undergo a coil-stretch transition under flow and we characterize the onset of elastic flow instability in a serpentine channel as a function of its curvature. The calibrated serpentine channel can then be used as a sensitive rheometer to detect even small normal stresses in unknown suspensions. Second, we employ a Y-channel, a powerful rheometer for measuring shear viscosities, to study the viscosity of active suspensions of e-coli bacteria. In this way we link the activity of the bacteria to the measured non-Newtonian effective viscosity for the first time.

Sunday, November 24, 2013 2:15PM - 4:25PM –

Session D1 Geophysical: Atmospheric II 323 - Michele Guala, University of Minnesota

2:15PM D1.00001 Influence of inflow condition on wind turbine operation and wake unsteadiness , KEVIN HOWARD, Saint Anthony Falls Laboratory, University of Minnesota, LEONARDO P. CHAMORRO, University of Illinois, MICHELE GUALA, Saint Anthony Falls Laboratory, University of Minnesota — A model wind turbine was tested in a closed-circuit wind tunnel under three different inflow conditions, (i) smooth wall turbulent boundary layer, (ii) preceding turbine wake and (iii) three dimensional sinusoidal hill wake, and three thermal stratifications. Two particle image velocimetry (PIV) fields were taken simultaneously upwind and downwind of the turbine along with the turbine voltage, which quantifies rotor fluctuations. Both wall-normal PIV fields were oriented on the centerline of the turbine and captured flow data in a window of approximately 1.1D by 1.1D, where D is the rotor diameter of the turbine. The upwind PIV measured the changing inflow conditions while both the voltage and downwind PIV field provided data that describes the response of the turbine and near wake to the inflow, respectively. Changes occurring in the inflow, whether upwind perturbation or thermal stability related, were statistically linked to the turbine voltage production and wake unsteadiness, as shown by turbulence intensity and swirling strength contours. A laboratory to field scale comparison is completed by inspecting light detection and ranging (Lidar) data taken upwind of the EOLOS utility scale, 2.5 MW wind turbine in conjunction with the turbine power production time signals.

2:28PM D1.00002 Stratificaiton Effects on wake of large wind turbines in wind farm[1] , KIRAN BHAGANAGAR, MITHU DEBNATH, University of Texas, San Antonio — The focus of the present talk is to demonstrate the interplay of the complex interactions between the wind turbulence and the wake turbulence under different stratification conditions. Large eddy simulation (LES) has been used to simulate flow over multi mega-watt wind turbines. The results have revealed different empirical relations for the mean velocity deficit decay and turbulence kinetic energy decay rates in the wake region of the wind turbine. The simulation for wind farm has revealed the wake decay rates as a function of the radial and streamwise distance from the upstream wind turbine. Vertical mixing plays a major role in altering the flow dynamics in the wake region.

[1]Support from NSF CBET-1348480, NSF HRD-1242180 and TACC Supercomputing

2:41PM D1.00003 Wind Tunnel Simulation of the Atmospheric Boundary Layer[1] , TRISTEN HOHMAN, ALEXANDER SMITS, LUIGI MARTINELLI, Princeton University — To simulate the interaction of large Vertical Axis Wind Turbines (VAWT) with the Atmospheric Boundary Layer (ABL) in the laboratory, we implement a variant of Counihan's technique [Counihan 1969] in which a combination of a castellated barrier, elliptical vortex generators, and floor roughness elements is used to create an artificial ABL profile in a standard closed loop wind tunnel. To examine the development and formation of the artificial ABL hotwire and SPIV measurements were taken at various downstream locations with changes in wall roughness, wall type, and vortex generator arrangements. It was found possible to generate a boundary layer at $Re_\theta \sim 10^6$, with a mean velocity that followed the 1/7 power law of a neutral ABL over rural terrain and longitudinal turbulence intensities and power spectra that compare well with the data obtained for high Reynolds number flat plate turbulent boundary layers [Hultmark et al. 2010].

[1]Supported by Hopewell Wind Power Ltd., and the Princeton Grand Challenges Program.

2:54PM D1.00004 Experimental investigations of a sphere anemometer: Wind tunnel and field tests[1] , HENDRIK HEISSELMANN, JOACHIM PEINKE, MICHAEL HOELLING, ForWind - University of Oldenburg — In our contribution we will compare the sphere anemometer and two standard sensors for wind energy and meteorology based on results from laboratory and atmospheric measurements. The sphere anemometer is a drag-based sensor for simultaneous wind speed and direction measurements. The new anemometer makes use of the velocity-dependent deflection of a lightweight sphere mounted on top of a flexible tube. The deflection of the sphere is detected by means of a highly sensitive light pointer, as used in atomic force microscopy. This allows for the detection of very small displacements and thus enables a high sensor resolution. In wind tunnel experiments the sphere anemometer, a 3D sonic anemometer and a standard cup anemometer were exposed to a turbulent wind field generated with a so-called active grid. All acquired data was compared to those of a highly resolving hot-wire probe. Moreover, the sphere anemometer and the two reference sensors were installed on two near-shore sites in the German Wadden Sea. Several month of data from these campaigns were analyzed regarding wind speed and direction measurements as well as durability and stability of the new anemometer.

[1]The presented work was founded by the German Ministry of the Environment, Nature Conservation and Nuclear Safety.

3:07PM D1.00005 MATERHORN Field Campaigns: An Overview[1] , HARINDRA FERNANDO, University of Notre Dame, ERIC PARDYJAK, University of Utah, MATERHORN TEAM[2] — Emerging important applications have resuscitated scientific and societal interests in mountain terrain flows. Funded by a MURI grant from the Office of Naval Research in 2011, the Mountain Terrain Atmospheric Modeling and Observations (MATERHORN) Program has achieved several important scientific milestones, which will be outlined in this presentation. MATERHORN has four principal thrusts - Modeling, Experimental, Technology and Parameterizations - that are symbiotically directed toward identifying model deficiencies and knowledge gaps, conducting process studies, and developing knowledge and tools for model improvements. Two comprehensive field studies were conducted during September 25 to October 31, 2012 (focusing on quiescent fair weather; wind speeds < 4 m/s) and May 1 to May 30, 2013 dealing with synoptic influence, moister surface conditions, and moderate (5 to 10 m/s) and strong (> 10m/s) wind periods. An unprecedented suite of high-end instrumentation was used, allowing investigations from tens of km to millimeters and hours to second scales. A host of phenomena that signifies flow interactions across a range of scales were identified, and the dynamics of a selected few will be discussed.

[1]Supported by ONR Grant # N00014-11-1-0709.
[2]www.nd.edu/ dynamics/materhorn

3:20PM D1.00006 Slope and Valley Flow Interactions in MATERHORN-1[1] , CHRISTOPHER M. HOCUT, R. DIMITROVA, Z. SILVER, Univ. of Notre Dame, S. DI SABATINO, Univ. of Notre Dame/Univ. of Salento, L.S. LEO, Univ. of Notre Dame, S.W. HOCH, Univ. of Utah, Y. WANG, U.S. Army Research Laboratory, E.R. PARDYJAK, Univ. of Utah, H.J.S. FERNANDO, Univ. of Notre Dame — In the fall 2012, the **M**ounta**i**n **Ter**rain Atmosp**h**eric M**o**deling and Obser**v**atio**n**s Program (MATERHORN) conducted its first extensive field experiment at the Granite Mountain Atmospheric Science Testbed (GMAST), US Army Dugway Proving Grounds (DPG), Utah. Of particular interest to MATERHORN-1 were the complex multi-scale interactions between thermally driven meso-scale up/down valley flows and up/downslope flows. To capture these phenomena, a suite of advanced instrumentation was used, which could identify and educe salient physical processes. LiDAR observations were particularly useful, showing the collision of the downslope flow with the valley flow, forming intense turbulent regions, intrusions and instabilities. To further investigate these intriguing interactions and identify meso-scale model shortcomings, WRF simulations have been conducted. In addition to the field measurements and computations, slope and valley flow interactions are the focus of an on-going laboratory study. The goal is to determine the nature of the interactions, determine if there are flow instabilities, examine the turbulence near the region of interaction, and develop a simple scaling in the flow destruction region.

[1]Research supported by Office of Naval Research Award N00014-11-1-0709.

3:33PM D1.00007 Effects of complex terrain on atmospheric flow: dividing streamline observations and quantification[1] , MICHAEL THOMPSON, HARINDRA FERNANDO, University of Notre Dame, SILVANA DI SABATINO, University of Notre Dame; University of Salento, LAURA LEO, University of Notre Dame, UNIVERSITY OF NOTRE DAME TEAM — As part of the MATERHORN field campaign on atmospheric flow in mountainous terrain, the dividing streamline concept for stratified flow over obstacles was investigated using smoke flow visualization and meteorological measurements. At small Froude numbers ($Fr < 1$), a stratified flow approaching a mountain either possesses enough kinetic energy to pass over the summit or else flow around the sides, with dividing streamlines separating the two scenarios. An isolated northwestern peak of the Granite Mountain, approximately $60m$ in height, was used for the study. Incoming flow velocities and temperature profiles were measured upstream using sonic anemometers and thermocouples mounted on a $32m$ tower, while onsite measurements were taken with portable weather stations. Sufficiently strong stratification was developed around 3:00AM GMT, with Froude numbers in the range for dividing streamlines to exist. In the first trial, suitably placed red smoke releases were used and in another trial white smoke was released from a $25m$ crane. In both cases well-defined dividing streamlines were observed and its vertical location was at a height about half of the mountain height, which is consistent with theoretical results based on Shepard's formula.

[1]This research was supported by the Office of Naval Research (ONR) grant number N00014-11-1-0709.

3:46PM D1.00008 The critical slope for orographic rain , ROBERT BREIDENTHAL, University of Washington, NEDJELJKA ZAGAR, University of Ljubljana — Krishnamurti has shown that orographic rain depends on the slope of the windward terrain rather than just the total elevation gain. A simple physical model is proposed to account for the effect of slope. Based on the inhibiting effect of vortex (rotational) acceleration on entrainment, a critical slope angle is derived. If the rate of orographic lifting is sufficiently large, the enhanced buoyancy from latent heat release increases the acceleration parameter. As a consequence, the entrainment rate of under-saturated air is reduced. The critical slope corresponds to the situation where the rate of condensation in a rising adiabatic parcel just equals the rate of evaporation due to the entrainment of under-saturated air. The model is also applied to the trigger conditions for towering cumulus in general.

3:59PM D1.00009 A numerical study of turbulent flow over complex aeolian dune fields: the White Sands National Monument , WILLIAM ANDERSON, Baylor University, MARCELO CHAMECKI, Penn State University, GARY KOCUREK, DAVID MOHRIG, The University of Texas at Austin — The structure and dynamics of fully-developed turbulent flows responding to aeolian dune fields are studied using large-eddy simulation with an immersed boundary method. An aspect of particular importance in these flows is the downwind migration of coherent motions associated with Kelvin-Helmholtz instabilities which originate at the dune crests. These instabilities are responsible for enhanced downward transport of high momentum fluid via the so-called turbulent sweep mechanism. However, the presence of such structures and their role in determining the bulk characteristics of fully developed dune field sublayer aerodynamics has received relatively limited attention. Moreover, many existing studies address mostly symmetric or mildly asymmetric dune forms. The White Sands National Monument is a field of aeolian gypsum sand dunes in southern New Mexico. In the dune field sublayer, the flow statistics resemble a mixing layer: at approximately the dune crest height, vertical profiles of streamwise velocity exhibit an inflection and turbulent Reynolds stresses are maximum; below this, the streamwise and vertical velocity fluctuations are positively and negatively skewed, respectively. We evaluate the spatial structure of Kelvin-Helmholtz instabilities present in the dune field sublayer – shear length, Ls, and vortex spacing, Lambda_x – and show that Ls = m Lambda_x, where m is approximately 8 in the different sections considered.

4:12PM D1.00010 Three-dimensional Quasi-Geostrophic Convection in the Rotating Cylindrical Annulus with Steeply Sloping Endwalls[1] , KEITH JULIEN, MICHAEL CALKINS, PHILIPPE MARTI, University of Colorado at Boulder — The rotating cylindrical annulus geometry was first developed by Busse (JFM 1970) as a simplified analogue for studying convection in rapidly rotating spherical geometries. Though it has provided a more tractable two-dimensional model than the sphere, it is formally limited to asymptotically small slopes and thus weak velocities in the direction parallel to the rotation axis. We present an asymptotically reduced three-dimensional equation set to model quasi-geostrophic convection in the annulus geometry where order one slopes are permissible; this model provides a closer analogue to quasi-geostrophic convection in spheres and spherical shells where steeply sloping boundaries are present. A linear stability analysis of the reduced equations shows that a new class of three-dimensional, convectively-driven Rossby waves is present in this system. The gravest modes exhibit strong axial variations as the slope of the boundaries becomes large. Additionally, higher-order eigenmodes showing increasingly complex axial dependence are found that possess critical Rayleigh numbers close to that of the gravest mode.

[1]NSF EAR CSEDI grant 1067944

Sunday, November 24, 2013 2:15PM - 4:25PM –

Session D2 Convection and Buoyancy-Driven Flows II: Heat Transfer 324 - Daniel Maynes, Brigham Young University

2:15PM D2.00001 Optimal aspect-ratio for heat transport in turbulent Rayleigh-Bénard convection in Cartesian geometry[1]

, KAI-LEONG CHONG, MATTHIAS KACZOROWSKI, KE-QING XIA, The Chinese University of Hong Kong — We present a three-dimensional direct numerical simulation study of the heat transfer efficiency, the Nusselt number Nu, as a function of the aspect ratio in turbulent Rayleigh-Bénard convection with Cartesian geometry. The study spans a range of the Rayleigh number Ra from 10^7 to 3×10^9 but at a fixed Prandtl number $Pr = 4.38$. A recent experimental and numerical study [1] has shown that the heat transfer efficiency increases significantly when the width of the convection cell is narrowed. In the present study, we carry out the simulations with even smaller aspect-ratio to further investigate the effect of cell confinement which is hard to achieve experimentally. It is found that there exists an optimal aspect ratio for heat transport at a given Ra. Furthermore we find an increase in the coherence of flow structures as the degree of confinement increases.

[1] S.-D. Huang, M. Kaczorowski, R. Ni, K.-Q. Xia. Confinement induced heat transport enhancement in turbulent thermal convection, submitted to Phys. Rev. Lett.

[1]This work was supported by the RGC of Hong Kong SAR (CUHK403811).

2:28PM D2.00002 Flow characteristics and heat transfer in wavy walled channels

, ZACHARY MILLS, TAPAN SHAH, VONTRAVIS MONTS, Georgia Institute of Technology, ALOK WAREY, SANDRO BALESTRINO, General Motors Global Research and Development, ALEXANDER ALEXEEV, Georgia Institute of Technology — Using lattice Boltzmann simulations, we investigated the effects of wavy channel geometry on the flow and heat transfer within a parallel plate heat exchanger. We observed three distinct flow regimes that include steady flow with and without recirculation and unsteady time-periodic flow. We determined the critical Reynolds numbers at which the flow transitions between different flow regimes. To validate our computational results, we compared the simulated flow structures with the structures observed in a flowing soap film. Furthermore, we examine the effects of the wavy channel geometry on the heat transfer. We find that the unsteady flow regime drastically enhances the rate of heat transfer and show that heat exchangers with wavy walls outperform currently used heat exchangers with similar volume and power characteristics. Results from our study point to a simple and efficient method for increasing performance in compact heat exchangers.

2:41PM D2.00003 Heat transfer and stability of horizontal convection with a moving forcing boundary[1]

, GREGORY SHEARD, TZEKIH TSAI, WISAM HUSSAM, KEAN YUNG WONG, Dept. of Mech. and Aero. Engineering, Monash University, MARTIN KING, Uni Klima, Bjerknes Centre for Climate Research, Norway — Horizontal convection describes a buoyancy-driven flow driven by a non-uniform supply of buoyancy across a horizontal forcing boundary achieved by a combination of heating and cooling. Horizontal convection establishes a horizontal flow in a thin boundary layer adjacent to the forcing boundary, and an overturning circulation is completed by way of a diffuse slow-moving return flow outside of the forcing boundary layer. Horizontal convection bears some similarity to global ocean currents, and so this fundamental study considers a second driving mechanism in conjunction with buoyancy, horizontal movement of the forcing boundary, as a model for wind-driven forcing on the flow. We characterize the combinations of Rayleigh number for buoyancy forcing and Reynolds number for mechanical forcing that produce three distinct regimes of behaviour: a forced-convection regime at high Reynolds numbers and low Rayleigh numbers, a mixed regime, and a free-convection regime dominated by Rayleigh number.

[1]Supported by the Australian Research Council through Discovery Grant DP120100153

2:54PM D2.00004 Local Wall Heat Flux[1]

, ROBERT KAISER, RONALD DU PUITS, University of Technology Ilmenau — Thermal convection is an omnipresent mechanism in nature and industry whereas its complexity is still a great challenge for scientists. A common model system to study natural thermal convection is the Rayleigh-Bénard setup. The flow inside a RB convection cell is driven by a temperature difference between top and bottom plate, while the heat loss throughout the sidewall is suppressed. A lot of effort has been taken to measure the global heat transport at high Ra spanning a wide Pr range. However, it is still unclear how it is locally distributed at the horizontal plates and how this distribution depends on the aspect ratio. We report local wall heat flux measurements using heat flux sensors at the surface of the heating plate. The measurements have been carried out in our large-scale RB experiment, called the "Barrel of Ilmenau" at $Ra = 4 \cdot 10^9$ varying $1 < \Gamma < 8$ and $Ra = 10^8$ varying $4 < \Gamma < 20$. Own measurements in a small rectangular RB cell shows that the time-average of the local heat flux at the surface of the plates can vary with respect to the position at the plate by about 30%. The locations of enhanced heat flux could be clearly associated with regions of strong plumeactivity like the area where plumes coming from the opposite plate and hit the plate surface.

[1]DFG PU 436/3-2

3:07PM D2.00005 Optically induced natural convection in a cylinder using conducting metal oxide films

, BRIAN J. ROXWORTHY, Department of Electrical & Computer Engineering, University of Illinois at Urbana-Champaign, KIMANI C. TOUSSAINT, SURYA P. VANKA, Department of Mechanical Science and Engineering, University of Illinois at Urbana-Champaign — We present a computational study of light- driven natural convection in a cylinder. We solve the coupled electromagnetic, heat transfer, and fluid mechanics equations in an axi-symmetric geometry with heating and fluid flow induced by optical absorption in a conducting metal oxide film comprised of Indium-Tin-Oxide (ITO). Calculations are performed as a function of the relevant optical input parameters including the wavelength of the illumination source (λ), the input power of the input light (P) which is assumed to have a Gaussian intensity distribution, and the numerical aperture of the focusing lens, defined as NA $= n\sin\theta$, where $n$is the index of refraction of the local medium and θ is the half-angle of the focused light cone. Due to the localized, spatially non-uniform illumination, fluid flow is induced for any finite Rayleigh number Ra > 0 and the resulting flow closely resembles a toroidal Rayleigh-Bénard convection pattern. The maximum fluid velocity scales linearly with Pand increases with increasing AR up to AR ~ 2; above this value, increasing h_{fluid} has no effect on the peak velocity. The optical actuation enables dynamic reconfigurability of the heating and convection patterns, which benefit lab-on-a-chip fluid mixing and particle manipulation.

3:20PM D2.00006 Wall to Wall Optimal Transport[1]

, GREGORY P. CHINI, University of New Hampshire, PEDRAM HASSANZADEH, Harvard University, CHARLES R. DOERING, University of Michigan — How much heat can be transported between impermeable fixed-temperature walls by incompressible flows with a given amount of kinetic energy or enstrophy? What do the optimal velocity fields look like? We employ variational calculus to address these questions in the context of steady 2D flows. The resulting nonlinear Euler–Lagrange equations are solved numerically, and in some cases analytically, to find the maximum possible Nusselt number Nu as a function of the Péclect number Pe, a measure of the flow's energy or enstrophy. We find that in the fixed-energy problem $Nu \sim Pe$, while in the fixed-enstrophy problem $Nu \sim Pe^{10/17}$. In both cases, the optimal flow consists of an array of convection cells with aspect ratio $\Gamma(Pe)$. Interpreting our results in terms of the Rayleigh number Ra for relevant buoyancy-driven problems, we find $Nu \leq 1 + 0.035Ra$ and $\Gamma \sim Ra^{-1/2}$ for porous medium convection (which occurs with fixed energy), and $Nu \leq 1 + 0.115Ra^{5/12}$ and $\Gamma \sim Ra^{-1/4}$ for Rayleigh–Bénard convection (which occurs with fixed enstrophy and for free-slip walls).

[1]This work was supported by NSF awards PHY-0855335, DMS-0927587, and PHY-1205219 (CRD) and DMS-0928098 (GPC). Much of this work was completed at the 2012 Geophysical Fluid Dynamics (GFD) Program at Woods Hole Oceanographic Institution.

3:33PM D2.00007 Solution breakdown due to natural convection of the boundary-layer radial flow on a constant temperature horizontal plate , RAMON FERNANDEZ-FERIA, CARLOS DEL PINO, ALBERTO FERNÁNDEZ-GUTIÉRREZ, Universidad de Málaga — The boundary-layer flow of a cold horizontal current exiting radially from a cylindrical vertical surface with a constant velocity over a hotter horizontal wall with constant temperature is analyzed. The temperature and velocity fields are coupled by buoyancy through the pressure gradients, so that the boundary-layer equations are made dimensionless with a radial characteristic length in which natural and forced convection become of the same order of magnitude, being the Prandtl number the only nondimensional parameter governing the problem. A similarity solution valid for the leading edge boundary-layer flow is obtained, yielding as a first order correction the effect of natural convection on Blasius' thermal boundary layer. This solution is also used to start the numerical integration of the equations to find out the location where the boundary-layer flow blows up due to the termination of the solution in a singularity. The physical nature of this singularity is analyzed and its position is characterized numerically. The heat flux from the horizontal wall up to this singularity is also characterized and qualitatively compared with previous experimental results from a related experimental setup.

3:46PM D2.00008 Convection to Sessile Droplets on Superhydrophobic Surfaces , DANIEL MAYNES, ROBB HAYS, JULIE CROCKETT, Brigham Young University — We report results from an investigation of the thermal convection to liquid droplets on heated horizontal superhydrophobic (SH) surfaces. We consider the transient response to droplets, initially at ambient temperature, as they are placed on heated SH surfaces at constant temperature. For comparative purpose we also consider the same scenario with smooth hydrophobic surfaces. The temporally varying droplet and surface temperatures were measured with an IR camera and a thermocouple, respectively. The droplets were also imaged with two CCD cameras and the time for the droplet to completely evaporate was monitored. For surface temperatures greater than the saturation temperature, high-speed video of the droplets was also acquired. Experiments were conducted over a range of surface temperatures varying from 40 to 215 C. The results show radically different behavior in the convection for the surface types considered. At all temperatures the total droplet evaporation time on the SH surfaces was significantly greater than on the smooth hydrophobic surface. At temperatures elevated above the saturation temperature the droplets on the SH surfaces remained at bulk temperatures significantly lower than the saturation temperature. Further, the droplets on the SH surfaces exhibited Leidenfrost-like behavior at surface temperatures far below the typical Leidenfrost point. Analysis of the data reveals overall heat transfer coefficients that decrease as the degree of superhydrophobicity increases.

3:59PM D2.00009 Anomalous convective heat transport and rain formation in cryogenic helium , K.R. SREENIVASAN, New York University, P. URBAN, P. HANZELKA, D. SCHMORANZER, Institute of Scientific Instruments ASCR, v.v.i., Královopolská 147, Brno, Czech Republic, L. SKRBEK, Faculty of Mathematics and Physics, Charles University, Ke Karlovu 3, 12116 Prague, Czech Republic — When a hot body A is thermally connected to a cold body B, the conventional wisdom is that heat flows from A to B. Here we describe the opposite case in which *heat flows from a colder but constantly heated body B to a hotter but constantly cooled body A through the thermal link of two-phase cryogenic helium*. Specifically, we provide experimental evidence that heat flows through liquid and gaseous layers of cryogenic helium from constantly heated but cooler bottom plate of the Rayleigh-Bénard convection cell to its hotter top plate that is constantly cooled. The bottom plate is heated uniformly and the top plate is cooled by heat exchange with liquid helium maintained at 4.2 K. Additionally, for certain experimental conditions, a rain of helium droplets is detected by small sensors placed in the cell interior at about half of its height. These results are expected to be of some consequence to laboratory studies of phase change in atmospheric clouds.

4:12PM D2.00010 An air curtain in the doorway of a ventilated space , DARIA FRANK, PAUL LINDEN, University of Cambridge — Air curtains are used to reduce the heat and the mass exchange between the indoor environment and the ambient. Their sealing ability is assessed in terms of the effectiveness E, the fraction of the exchange flow prevented by the air curtain compared to the open-door situation. Previous work studied the air curtain effectiveness when the doorway is the only means of ventilating a space. In this talk we examine effects of an additional displacement ventilation pathway on the effectiveness. The main controlling parameter is the deflection modulus D_m which is the ratio between the momentum flux of the air curtain and the transverse forces due to the displacement ventilation. For small values of D_m the air curtain is drawn inside the space by the ventilation flow. For high values of D_m the flow is controlled by the air curtain. A smooth transition occurs between these two regimes and we estimate the D_m value for the onset of this transition. Our model makes a quantitative prediction of $E(D_m)$ in the ventilation-driven regime, and explains qualitatively the shape of the curve in the other two regimes. Laboratory experiments were conducted to test the proposed model. The experimental data were compared to theoretical predictions and good agreement was found.

Sunday, November 24, 2013 2:15PM - 4:25PM –

Session D3 Multiphase Flows II 325 - John R. Saylor, Clemson University

2:15PM D3.00001 Surface wetting effects on drop passage through a confining orifice[1] , ANKUR BORDOLOI, ELLEN LONGMIRE, Aerospace Engineering and Mechanics, University of Minnesota — The motion of gravity-driven drops (Bo $\sim$ 2-10) through a sharp-edged confining orifice is studied in a liquid/liquid system for both hydrophobic (HPB) and hydrophilic (HPL) orifice surfaces. The drop interface is tracked by high-speed imaging, and fluid velocity fields are obtained by PIV. When a drop impacts the leading edge of the orifice, the drop fluid contacts the solid surface immediately, and the resulting interfacial contact lines begin propagating away from the edge. The final drop outcome (capture, release or break-up) is influenced by the motion of the contact lines as well as the contact force between the drop fluid and the orifice surface. In the HPB case, the contact line motion is limited, and the contact force acting against drop passage is weak. In the HPL case, the contact line motion strongly inhibits drop passage by spreading fluid across the upper surface of the orifice plate. For drops that break into multiple volumes, the wettability influences both the break-up location and fractional volume of the resulting satellite drop.

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

2:28PM D3.00002 Simulations of Three-dimensional Droplet Deformation in a Square-Duct at Moderate Reynolds numbers[1] , JEREMY HORWITZ, PURUSHOTAM KUMAR, PRATAP VANKA, University of Illinois at Urbana-Champaign — We present results of numerical simulations of deformation of a confined droplet in a three-dimensional square-duct flow using a multiphase Lattice Boltzmann Method. We have studied the effects of capillary number, Reynolds number, and viscosity ratio on the droplet deformation characteristics. Unlike in the Stokes' limit where deformation is governed by a competition between viscous shear and interfacial tension, at higher Reynolds numbers, inertial effects play an increasingly important role. We observe that the deformation history is non-monotonic and contains an overshoot before relaxing to a steady deformed state. In contrast, the capillary number is seen to affect the magnitude of the deformation history and the time at which the peak deformation occurs. The viscosity ratio has a relatively modest effect on the magnitude of the deformation compared with the effects of Reynolds and capillary numbers. However, compared with the Reynolds number, the viscosity ratio and capillary number have a significant effect on the time to reach a steady state.

[1]University of Illinois at Urbana-Champaign, Air Conditioning and Refrigeration Center

2:41PM D3.00003 A Three-dimensional Numerical Study of Immiscible Droplet Deformation in a right angle bend , PURUSHOTAM KUMAR, JEREMY HORWITZ, SURYA VANKA, University of Illinois at Urbana-Champaign — We present a numerical study of deformation of an immiscible droplet in a right angle bend. We have used volume-of-fluid method to track the interface and variable density Navier-Stokes equations to solve for the flow field. A second-order accurate fractional step algorithm is used to integrate the equations. The VOF is also coupled to a level-set method to get a smoothed interface shape for surface tension calculations. We study the effects of density and viscosity ratios (between droplet and carrier fluids), Reynolds number, Capillary number and aspect ratio between droplet and duct size on the deformation characteristics. We investigate the elongation of the droplet in axial direction and the stretching or contraction of the droplet in the lateral direction. Depending on the value of above mentioned parameters droplet can take different shapes, namely, spherical, bullet and parachute. At moderately higher Reynolds numbers we also observe satellite droplet breaking from the original droplet.

2:54PM D3.00004 Breakup of an oil slick mixed with dispersants by breaking wave[1] , CHENG LI, ANNE HOLSER, JOSEPH KATZ, Johns Hopkins University — After oil spill, coherent oil slick are entrained by breaking ocean waves together with air, which produces a complicated three-phase flow, involving a wide range of length and time scales. The oil droplet size distribution is a crucial factor affecting the physical and chemical dispersion of oil spills, but little is known about oil droplet formation mechanism and droplet size distributions during and immediately after the impact of breaking waves. In our experimental study, we investigate the breakup of an oil slick in a specialized wave tank. The widely used dispersant Coexist 9500-A at different dispersant to oil ratio is used for varying the surface tension of crude oil (MC252 surrogate) in the 10^{-1} to 10 mN/m range. The dispersant is applied either by premixing or surface spraying, the latter consistent with typical application. The results include high-speed images of the oil and bubbles' entrainment, showing the resulting formation of a series of droplet clouds during multiple "plunges" associated with a single propagating breaking wave. High-speed inline digital holographic cinematography is employed to quantify the oil droplet size distribution, and the impact of droplet-bubble interactions on the entrainment process for varying Weber numbers, and wave properties, from spilling to plunging breakers.

[1]Supported by Gulf of Mexico Research Initiative (GoMRI)

3:07PM D3.00005 Inkjet Printer Drop Impact on Coated and Uncoated Papers , SAMAN HOSSEINI, RAFAEL ORSI KOGA, NASSER ASHGRIZ, SANJEEV CHANDRA, University of Toronto — The impact of ink drops generated by a solid ink inkjet printer on coated and uncoated papers were investigated. Ink drops are different impact velocities and different initial temperatures were tested. A Xerox solid ink inkjet is used in this study. In this printer, the solid ink is heated to temperatures of about 90C and then ejected out of the printed. The ink drop solidifies as soon as it impact on the paper. Small, about 39 micron droplets were impacted on a paper positioned 0.5 mm from the print head. SEM images of individual drops were obtained and analyzes to determine the droplet spread diameter, droplet height, and droplet contact angles. In addition, the texture and shape of the drops were categorized according to the impact parameters. There is a substantial difference between the droplet impact results for coated versus uncoated papers. In addition, the temperature of the substrate at the time of impact made a significant difference, since small drops cool very fast and they do not attach properly to the paper at room temperatures.

3:20PM D3.00006 Tribonucleation of bubbles , SANDER WILDEMAN, HENRI LHUISSIER, CHAO SUN, University of Twente, ANDREA PROSPERETTI, Johns Hopkins University, DETLEF LOHSE, University of Twente — We report on the nucleation of bubbles on solids that are gently rubbed against each other while immersed in a gas-supersaturated liquid. For given supersaturation and surface material, bubble nucleation is only observed beyond a certain threshold for the rubbing force and velocity. Above this threshold, a regularly spaced row of growing bubbles is left behind on the surface. Direct observation through the bottom of a transparent solid shows that each bubble in the row results from the early coalescence of several microscopic bubbles. From a detailed study of the wear tracks it seems that these gas nuclei originate from a local fracturing of the surface asperities in the contact area.

3:33PM D3.00007 Stability analysis applied to the early stages of viscous drop breakup by a high-speed gas stream[1] , JUAN C. PADRINO[2], Los Alamos National Laboratory, ELLEN K. LONGMIRE, University of Minnesota — The instability of a liquid drop suddenly exposed to a high-speed gas stream behind a shock wave is studied by considering the gas-liquid motion at the drop interface. The discontinuous velocity profile given by the uniform, parallel flow of an inviscid, compressible gas over a viscous liquid is considered, and drop acceleration is included. Our analysis considers compressibility effects not only in the base flow, but also in the equations of motion for the perturbations. Recently published high-resolution images of the process of drop breakup by a passing shock have provided experimental evidence supporting the idea that a critical gas dynamic pressure can be found above which drop piercing by the growth of acceleration-driven instabilities gives way to drop breakup by liquid entrainment resulting from the gas shearing action. For a set of experimental runs from the literature, results show that, for shock Mach numbers ≥ 2, a band of rapidly growing waves forms in the region well upstream of the drop's equator at the location where the base flow passes from subsonic to supersonic, in agreement with experimental images. Also, the maximum growth rate can be used to predict the transition of the breakup mode from Rayleigh-Taylor piercing to shear-induced entrainment.

[1]The authors acknowledge support of the NSF (DMS-0908561).
[2]This work was conducted when the first author was affiliated with the University of Minnesota.

3:46PM D3.00008 Break-up of droplets in a concentrated emulsion flowing through a narrow constriction[1] , SINDY TANG, LIN FAN, YUNHAN CHEN, LIAT ROSENFELD, Stanford University — We describe the break-up of droplets in a concentrated emulsion during its flow as a 2D monolayer in a microchannel consisting of a narrow constriction. Analysis of the behavior of over 4000 drops shows that the number of break-ups increases with increasing flow rate, entrance angle to the constriction, and size of the drops. As single drops do not break at the highest flow used, break-ups arise primarily from droplet-droplet interactions. Droplet-droplet interactions are stochastic; they cause fluctuations in the local strain rate and deformation each drop experiences. Analysis of droplet properties at a temporal resolution of 10 microseconds makes it possible to relate drop deformation with break-up probability. Similar to previous studies on single drops, no break-up is observed below certain critical flow rates and droplet deformations. Unlike previous studies, however, not all drops break above the critical values. Instead, the probability of break-up increases with flow rate and the local deformation of the drops.

[1]We acknowledge funding from the Stanford Woods Institute for the Environment, the California Sea Grant Project through NOAA's National Sea Grant College Program, and the Donors of the American Chemical Society Petroleum Research Fund.

3:59PM D3.00009 Droplet impact on falling liquid films[1] , OMAR MATAR, ZHIZHAO CHE, IVAN ZADRAZIL, GEOFFREY HEWITT, CHRISTOS MARKIDES, Imperial College London — Droplet impact is a ubiquitous phenomenon in nature, and has a wide range of applications; these include inkjet printing, spray painting, and surface cleaning. In this study, we examine the impact of droplets on falling liquid films, which is an event that occurs in various two-phase flows, such as annular flows and spray cooling. High-speed photography is used to visualise droplet impact, and associated phenomena, on a uniform falling liquid film, which is created on a flat substrate with controllable thickness and flow speed. Different phenomena are observed and analysed for droplet impact at different impact speeds, angles, and film thicknesses and flow speeds. The results of the present work are part of a programme to elucidate the complex dynamics of multiphase flows and to develop validated numerical tools for accurate predictions.

[1]EPSRC Programme Grant EP/K003976/1

4:12PM D3.00010 Theoretical and numerical investigation of turbulence/interface interactions due to surface tension effects , JEREMY MCCASLIN, CHIAN YEH GOH, OLIVIER DESJARDINS, Cornell University — The interaction between turbulence and an interface subjected to surface tension is studied by inserting an interface into a triply periodic box of decaying homogeneous isotropic turbulence, simulated with a volume-of-fluid scheme on a mesh of size 512^3. Unity density and viscosity ratios are used in order to isolate the interaction between turbulent eddies and the surface tension force. Interfacial height correlations are used to study the spatial scales of corrugations on the interface. At a Taylor-microscale Reynolds number of $\mathrm{Re}_\lambda = 146$, a case with zero surface tension is first considered, yielding a passive interface that moves materially with the fluid. Simulation results confirm a theoretically predicted universal κ^{-2} scaling of the corrugation power spectral density, where κ is the wavenumber. In the presence of surface tension, the corrugation spectrum follows the κ^{-2} law for large scales, but then deviates towards a $\kappa^{-11/3}$ scaling once inertia becomes balanced by surface tension. Coupling between the interface and surrounding fluid modulates the turbulent flow, and a transfer of turbulent kinetic energy from low to high wavenumbers is observed in the energy spectrum.

Sunday, November 24, 2013 2:15PM - 4:25PM –

Session D4 DFD Minisymposium: Nanobubbles 326 - Detlef Lohse, University of Twente

2:15PM D4.00001 Experimental Studies of Nanobubbles at Solid-Water Interfaces[1] , XUEHUA ZHANG, University of Melbourne — When a hydrophobic substrate is in contact with water, gas bubbles thinner than 100 nm can form at the interface and stay for long time under ambient conditions. These nanobubbles have significant influence on a range of interfacial processes. For example, they give rise to hydrodynamic slip on the boundary, initiate the rupture of thin liquid films, facilitate the long-ranged interactions between hydrophobic surfaces, and enhance the attachment of a macroscopic bubble to the substrate. Experimentally, it is nontrivial to characterize such small fragile bubbles and unravel their fundamental physical properties. Based on our established procedure for the nanobubble formation, we have systematically studied the formation, stability and response of nanobubbles to external fields (e.g. sonication, pressure drop and temperature rise). By following the bubble morphology by atomic force microscopy, we show that the loss or gain of the nanobubble volume is achieved mainly by the change in the bubble height. The pinning on the three-phase boundary has significant implication on the properties of nanobubbles under various conditions. This talk will cover the effects of the substrate structures on the nanobubble formation, and the response of nanobubbles to the gas dissolution, the temperature increase, the extended gentle ultrasound or the substantial pressure drop in the environment.

[1]We acknowledge the support from Australian Research Council (FFT120100473).

2:41PM D4.00002 Dynamic equilibrium explanation for nanobubbles unusual temperature and saturation dependence , L. GARY LEAL, UCSB — Recent experimental evidence demonstrates that nanobubbles exhibit unusual behavior in response to changes in temperature and gas saturation in the liquid, an observation that may shed light on the mysterious origin of their stability. In this talk, we discuss an alternate formulation of the dynamic equilibrium mechanism for nanobubbles that predicts rich behavior in agreement with these measurements. Namely, we show that stable nanobubbles exist in narrow temperature and dissolved gas concentration ranges, that there is a maximum and minimum possible bubble size, and that nanobubble radii decrease with temperature. We also discuss these predictions in the context of other current hypotheses for nanobubble stability such as the recently-proposed diffusive "traffic jam" model.

3:07PM D4.00003 A new theory of bubble stability: Implications for nanobubbles at surfaces and in bulk solution[1] , VINCENT CRAIG, Australian National University — Nanobubbles on hydrophobic surfaces can be imaged using Atomic Force Microscopy and are implicated in the very long-range attraction measured between hydrophobic surfaces. However, the widely accepted theory of bubble dissolution predicts that small bubbles under the influence of Laplace pressure should rapidly dissolve resulting in bubble lifetimes of less than a second.[2] Such short lifetimes should preclude nanobubbles from having an effect on surface force measurements or being observed by AFM,[3] yet nanobubbles are readily observed by AFM and widely implicated in force measurements between hydrophobic surfaces. This has led to a number of attempts at describing their unexpected stability, though no explanation is currently widely accepted. Additionally, nanobubbles have contact angles substantially greater (measured through the more dense liquid phase) than the equivalent macroscopic contact angle. It is clear that nanobubbles at surfaces pose a number of problems that are yet to be resolved. Additionally, recent reports of long-lived nanobubbles in bulk solution add to the mystery. Here we present a new theory describing the stability of nanobubbles. We calculate their lifetimes as a function of gas supersaturation and explain the long lifetimes observed. The same theory predicts that bulk nanobubbles should be stable under certain circumstances. Further, in an extension of this work we explain the difference in contact angle between the nanoscopic and macroscopic measurements and describe in detail the process by which nanobubbles are formed during solvent exchange. Experimental evidence is presented supporting this new approach and showing that this theoretical framework has parallels in other nucleated systems.

[1]This work was supported by an ARC Future Fellowship.

[2]Epstein, P. S.; Plesset, M. S., *Journal of Chemical Physics* **1950,** *18* (11), 1505-1509.

[3]Ljunggren, S.; Eriksson, J. C., *Colloids and Surfaces a-Physicochemical and Engineering Aspects* **1997,** *130*, 151-155.

3:33PM D4.00004 Surface nanobubbles: Theory, numerics and experiments , JOOST H. WEIJS, Physics of Fluids, University of Twente — When a solid is brought into contact with water, surface nanobubbles can be formed at the solid-liquid interface. These nanobubbles are small; their height is of order 10nm and their lateral sizes vary from 10-100 nm. Initially, the only proof of the existence of surface nanobubbles was delivered by atomic force microscopy. Later, additional techniques such as infrared attenuated total reflectance have confirmed the existence of gaseous domains on the solid-liquid interface. Before this overwhelming evidence, the existence of surface nanobubbles was controversial because they possess some unusual properties. For example, nanobubbles are surprisingly robust against dissolution by diffusion and Laplace pressure: Instead of the expected lifetime of about a microsecond, nanobubbles are found to survive for several hours and in some cases even several days. Additionally, surface nanobubbles are flatter than predicted by Young's law and are able to resist strong tensile stresses ($\sim$-6 MPa), rather than serving as a nucleation site for a macroscopic bubble. A deep understanding of surface nanobubbles is crucial for practical applications (*e.g.* drag reduction in microfluidic devices) but nanobubbles also pose fundamental questions on the validity of continuum models at the nanoscale. In this talk, we will discuss these open questions in detail by considering theoretical efforts and molecular dynamics simulations. Theoretically, we study the consequences of a pinned contact line. We find that the pinned contact line can explain the long lifetimes and many other nanobubble properties. From molecular dynamics results, we clarify the influence of the gas species on the contact angle. Finally, we will discuss some very recent experimental and theoretical work on the effects of an acoustic field on nanobubbles. We provide experimental data combined with a theoretical analysis and find that the acoustic driving can cause the nanobubbles to grow by rectified diffusion.

3:59PM D4.00005 A theory for metastabilities in bubble nucleation: can it help explaining nanobubbles?[1] , CARLO MASSIMO CASCIOLA, Dipartimento di Ingegneria Meccanica e Aerospaziale, Università di Roma La Sapienza — The stability and the very existence of nanobubbles on a solid-liquid interface is a conundrum that has been puzzling the community of researchers working in the field since their discovery through AFM measurements in the late nineties. Nanobubbles are typically flat, with height on the order of 5-10 nm and lateral size order 100 nm or less. Pinning of the contact line presumably plays a crucial role and, based on classical estimates, they should dissolve almost immediately while they are instead reported to persist for days. Recently we developed a novel theoretical approach that is able to predict the heterogeneous nucleation path, and to explain the catalytic effect of geometrical defects in lowering the associated free-energy barrier (Giacomello et al., PRL 2012). The theory bridges the scales from nanometer to micron, and is then suitable for dealing with nanobubbles, as shown by comparison with advanced rare-event techniques used to evaluate the metastability in the atomistic context (Giacomello et al., Langmuir 2012). The interest of the approach is that it can provide an estimate for the transition frequency, i.e. the average lifetime of a metastable configuration. As will be discussed, the model can in principle be enriched to account for the interaction of the gas phase with the solid, indicated as responsible for the almost universal contact angle observed in the nanobubbles (Weijs et al., PRL 2012). If nanobubbles can be explained in the context of equilibrium statistical ensembles, as long-lived metastable states associated with a complex free-energy landscape, the work under way could shed new light on the elusive subject of their persistence. At present we cannot however exclude substantial non-equilibrium effects, outside the concept of metastability in the strict statistical-mechanics sense and associated, e.g., with thermal gradients.

[1]ERC 2013 is acknowledged for support

Sunday, November 24, 2013 2:15PM - 4:25PM –

Session D5 CFD II: LES I 327 - Satbir Singh, Carnegie Mellon University

2:15PM D5.00001 Direct Simulations of Breaking Ocean Waves with Data Assimilation[1] , JAMES ROTTMAN, DOUGLAS DOMMERMUTH, LUCAS RHYMES, Science Applications International Corporation — An algorithm is developed to assimilate ocean wave data into the Numerical Flow Analysis (NFA) code. NFA is a Cartesian-based implicit LES code with Volume of Fluid (VOF) interface capturing. The assimilation of data into NFA permits the investigation of higher bandwidths than is possible using either High-Order Spectral (HOS) methods or field measurements. NFA models wave breaking that cannot be modeled using HOS. Direct simulations of wave breaking allow a more detailed analysis of ocean-wave physics than is possible using remote sensing of the ocean surface. Examples of simulated ocean surface waves exhibiting wave breaking, based on the JONSWAP or Pierson-Moskowitz ocean wave spectra, will be shown. Future plans include performing data assimilation of larger patches of the ocean surface with higher resolution to investigate the statistics and underlying structure of breaking waves.

[1]Funded by Dr. Thomas Drake at the Office of Naval Research (contract number N00014-12-C-0568)

2:28PM D5.00002 Large-eddy simulation of boundary layer flow on a non-uniform grid using explicit filtering and reconstruction , LAUREN GOODFRIEND, FOTINI KATOPODES CHOW, University of California, Berkeley, MARCOS VANELLA, ELIAS BALARAS, George Washington University — Many realistic flows, such as the urban boundary layer, are too expensive to simulate directly. Large-eddy simulation (LES) and adaptive mesh refinement (AMR) reduce the computational cost of turbulence modeling by restricting resolved length scales, but combining these techniques generates additional errors. The grid refinement interfaces in AMR grids can reflect resolved energy and create interpolation errors. This study investigates the use of explicit filtering and reconstruction to mitigate grid interface errors in LES of a pressure gradient forced boundary layer. The domain is split in the streamwise direction into two equally sized structured grids, one fine and one coarse, with periodic boundaries in the streamwise and spanwise directions. This simple test case allows observation of the effects of the grid interfaces. Explicit filtering is found to reduce accumulation of resolved energy at the fine-to-coarse interface and improve the shape of coherent structures, compared to basic LES. Reconstruction of the subfilter velocity is shown to further the improvements of explicit filtering. These results inform the use of LES on block-structured non-uniform grids, such as nested grids in local atmospheric models or on more complex Cartesian AMR grids.

2:41PM D5.00003 Large Eddy Simulations of Turbulent Reacting Flows in an Opposed-Piston Two Stroke Engine , SHALABH SRIVASTAVA, HAROLD SCHOCK, FARHAD JABERI, Michigan State University — The two-phase filtered mass density function (FMDF) subgrid-scale model has been used for large eddy simulation (LES) of turbulent spray combustion in a generic single cylinder, opposed-piston, two-stroke engine configuration. The LES/FMDF is implemented via an efficient, hybrid numerical method in which the filtered compressible Navier-Stokes equations are solved with a high-order, multi-block, compact differencing scheme, and the spray and FMDF are implemented with stochastic Lagrangian methods. The reliability and consistency of the numerical methods are established for the engine configuration by comparing the Eulerian and Lagrangian components of the LES/FMDF. The effects of various operating conditions like boost pressure, heat transfer model, fuel spray temperature, nozzle diameter, injection pressure, and injector configuration on the flow field, heat loss and the evolution of spray and combustion are studied.

2:54PM D5.00004 Scalable FDF Simulation of Reacting Flows , PATRICK H. PISCIUNERI, S. LEVENT YILMAZ, University of Pittsburgh, PETER A. STRAKEY, National Energy Technology Laboratory, MEHDI B. NIK, PEYMAN GIVI, University of Pittsburgh — The "irregularly portioned Lagrangian Monte Carlo-finite difference" (IPLMCFD) methodology is developed for efficient large eddy simulation (LES) via the filtered density function (FDF) subgrid scale closure. This methodology is particularly suited for simulation of chemically reacting flows and offers efficient utilization of thousands of processors. Various aspects of the scalability are presented for the LES of several premixed and non-premixed turbulent flames at low and high speeds. This method paves the way for petascale LES/FDF.

3:07PM D5.00005 Evaluation of the Partially-Averaged Navier-Stokes method for high Mach flows , 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 but not for the high speed flows. This will be demonstrated in the present work. Furthermore, some of the latest variants of the PANS models will be discussed as well. For the present study, we use the PANS ζ-f model which is based on the near-wall RANS ζ-f model. This RANS model is a variant of the $v^2 - f$ model, the difference being that a transport equation for the wall-normal velocity scale ratio ζ is included rather than one for the velocity scale. Additionally, an elliptic relaxation equation for f, which is a parameter closely related to the pressure strain redistribution term, is solved. It is expected that such PANS variant could accurately predict a flow considered here. A test case is a transonic flow over a channel bump. It has been used as a benchmark for turbulence modelling in shock-wave/boundary-layer interactions. Measurements, but also previous RANS calculations, are used as a reference point to the present calculations.

3:20PM D5.00006 Large-eddy simulations of impinging jets at high Reynolds numbers , WEN WU, UGO PIOMELLI[1], Queen's University, Kingston, Ontario, Canada — We have performed large-eddy simulations of an impinging jet with embedded azimuthal vortices. We used a hybrid approach in which the near-wall layer is modelled using the RANS equations with the Spalart-Allmaras model, while away from the wall Lagrangian-averaged dynamic eddy-viscosity modelled LES is used. This method allowed us to reach Reynolds numbers that would be prohibitively expensive for wall-resolving LES. First, we compared the results of the hybrid calculation with a wall-resolved one at moderate Reynolds number, $Re = 66,000$ (based on jet diameter and velocity). The mean velocity and Reynolds stresses were in good agreement between the simulations, and, in particular, the generation of secondary vorticity at the wall and its liftup were captured well. The simulation cost was reduced by 86%. We then carried out simulations at $Re = 266,000$ and 1.3 million. The effect of Reynolds number on vortex development will be discussed.

[1]Canada Research Chair in Computational Turbulence, HPCVL-Sun Microsystems Chair in Computational Science and Engineering

3:33PM D5.00007 Impact of Model Fidelity on Jet Impingement Simulations , BENJAMIN REIBMAN, MIKE BENSON, United States Military Academy, DAVID HELMER, GREGORY RODEBAUGH, GE — Turbulence modeling in the RANS framework has difficulty properly capturing free jet impingement, as both the shear layer of the jet upon entrance to the free stream and the impingement onto the flat plate are complex turbulent situations. In an effort to assess the accuracy of different turbulence modeling approaches and quantify fidelity/computational cost tradeoffs, a case study was conducted using the experimental data of Cooper et al 1993. This study consisted of both RANS and LES modeling. RANS analysis was conducted over a sweep of case conditions (z/D = 2, 6 and Re = 23000, 70000) leveraging multiple commercial solvers and a variety of common two-equation turbulence models. A grid refinement study was conducted with both structured and unstructured meshes to determine grid and solver dependence. The predictive capability of DDES in addition to WALE and dynamic one-equations k LES SGS closures for the z/D = 2 Re = 23000 case were compared against RANS and experimental data. The experimental data consisting of Nusselt numbers over radial diameters, pipe flow bulk velocity and wall jet velocity were the principal means of determining accuracy of the various models. This study will provide an assessment of the impact of the use of lower fidelity models in future analysis involving more complex geometries. This will provide guidance on the required fidelity at each stage of the design process in multiple fields utilizing impinging jets.

3:46PM D5.00008 Simulations of a Normal Shock Train in a Constant Area Duct Using Wall-Modeled LES , ZACHARY VANE, IVAN BERMEJO-MORENO, SANJIVA LELE, Stanford University — Large-Eddy Simulations (LES) of a turbulent boundary layer interacting with a normal shock train in a constant area duct (STCAD) are performed using an unstructured solver. Comparisons between wall-modeled LES (WMLES) and wall-resolved LES (WRLES) calculations of a spanwise-periodic flow at M=1.61 and Re = 16,200 are used to evaluate an equilibrium wall-model's ability to replicate the wall-resolved results. The WMLES approach is then used for simulations at the flow conditions (M=1.61, Re=162,000) of the Carroll & Dutton STCAD experiments where traditional WRLES was inaccessible. Spanwise-periodic WMLES calculations were unable to duplicate the experimental wall pressure and Laser Doppler Velocimetry data obtained along the spanwise center plane of the duct. Investigations of the full, low aspect ratio duct geometry were then performed using WMLES. Comparisons with experimental data provide an assessment of the wall-model's ability to simulate realistic, high Reynolds number, non-equilibrium flows. However, the lack of information with respect to the sidewall boundary layers from the experiment led to a WMLES parameter study of the effects of boundary layer confinement on the shock train. Initial results suggest that the tunnel blockage due to the boundary layer displacement thickness determines many of the STCAD's characteristics. A possible improvement to the wall-model through the inclusion of previously omitted non-equilibrium terms is currently being pursued.

3:59PM D5.00009 CFD predictions of confined turbulent swirling flows in a microscale multi-inlet vortex reactor , MICHAEL OLSEN, Iowa State University, YANXIANG SHI, Massachusetts Institute of Technology, GIANLUCA IACCARINO, Stanford University, RODNEY FOX, Iowa State University — Turbulent swirling flows have proven to be an efficient way of promoting mixing for chemical reactions. Accordingly, a multi-inlet vortex reactor was designed for use in the synthesis of nanoparticles. LES and RANS simulations have been applied to understanding the underlying fluid dynamics in this reactor. Both simulations are performed with the open-source platform, OpenFOAM, and the validity of the chosen models are verified against μ-PIV data. For the LES framework, the simplest and yet the most commonly chosen model, Smagorinsky-Lilly model, is used. For the RANS simulations, however, the k–ε model is not capable of capturing the swirling motions. Instead, the four-equation v^2–f model is formulated to account for the velocity fluctuations perpendicular to the streamlines and is therefore chosen in this work. Comparisons of the simulation results with the experimental data show both approaches accurately predict the mean velocity fields. Considering the computational cost, the RANS with the v^2–f model is recommended for obtaining statistical quantities whereas the LES simulations are more suitable for understanding transient flow behaviors. Based on the validation, the velocity field as well as the turbulence field is also analyzed.

4:12PM D5.00010 Large Eddy Simulation for round jet in cross-flow using Local Mesh Refinement , MEHTAP CEVHERI, THORSTEN STOESSER, Cardiff University — The aim of this research is the simulation of near field multi-phase plumes in cross-flows to understand the physical processes of oil spill in Gulf of Mexico. Since this is a multi-phase and multi-scale problem, a local mesh refinement (LMR) technique has been coupled to the multi-grid method to solve the unsteady, incompressible Navier-Stokes problem on a Cartesian grid with staggered variable arrangement. Wall-Adapting Local Eddy Viscosity (WALE) subgrid model has been used to simulate the turbulent flow. In this current study, the verification of the developed code will be presented before the simulation of multi-phase plumes. The accuracy of local mesh refinement and the subgrid model are presented with two test cases: moderate Reynolds number turbulent channel flow and a round turbulent jet into a laminar cross-flow. For the first test case, turbulence statistics for the fully developed turbulent flow are compared with the DNS data. For the second test case, a simulation with a 3.3 velocity ratio and 6930 jet Reynolds number is tested and compared with the experimental and other computational data.

Sunday, November 24, 2013 2:15PM - 4:25PM –

Session D6 Microfluids: Flow in Microchannels 328 - Soojung Claire Hur, Harvard University

2:15PM D6.00001 ABSTRACT WITHDRAWN –

2:28PM D6.00002 Measuring the thermal diffusion coefficients of artificial and biological particles in a microfluidic chip[1] , CHAO ZHAO, ALPARSLAN OZTEKIN, XUANHONG CHENG, Lehigh University — Particle thermophoresis refers to the migration of colloids under a temperature gradient. The thermophoretic velocity is proportional to the particle thermal diffusion coefficient and temperature gradient. However, in the literature, there are discrepancies about the mechanism for thermal diffusion and the reported values of the thermal diffusion coefficients are inconsistent for comparable systems. Furthermore, the thermal diffusion behavior of biological vesicles is underinvestigated. Here an optical method based on capillary is presented to measure the thermal diffusion coefficients of artificial and biological particles. By applying a temperature gradient along the width of the capillary, net velocity of microparticles and fluorescent intensity redistribution of nanoparticles are quantified to derive the thermal diffusion coefficients. The thermal diffusion coefficients of polystyrene beads as well as pseudoviral particles in physiological solutions are obtained in our work, and the values are compared with those from the literature. The differences are discussed in terms of interfacial interactions. This study provides insight into the transport of biological particles in a thermal gradient and will aid the design of separation devices.

[1]NIAID-1R21AI081638

2:41PM D6.00003 The clogging cascade of an array of microchannels , ERIN BARNEY, EMILIE DRESSAIRE, Trinity College, HOWARD STONE, Princeton University — The manipulation and filtration of dilute suspensions of microparticles are important processes for both natural and engineered systems. Relying on the comparable lengthscales of the microchannels and microparticles, these systems are particularly susceptible to blockage. Studies at the single-pore level have established that the clogging of a microchannel is controlled by colloidal and hydrodynamic interactions. However, clogging is a multiscale process; the formation of single-pore level clogs often results in the blockage of a macroscopic system. The dynamics of this series of clogging events or clogging cascade are studied here. We investigate the blockage of an array of parallel microchannels and show in particular, that the rate of clog formation decreases during the clogging cascade. Through experimental measurements and theoretical analysis, we demonstrate the roles of colloidal and hydrodynamic effects in the dynamics of the clogging cascade.

2:54PM D6.00004 Dissipative particle dynamics modeling of blood flow in arterial bifurcations[1] , XUEJIN LI, Division of Applied Mathematics, Brown University, Providence, RI 02912, USA, KIRILL LYKOV, IGOR V. PIVKIN, Institute of Computational Science, University of Lugano, Lugano 6904, Switzerland, GEORGE EM KARNIADAKIS, Division of Applied Mathematics, Brown University, Providence, RI 02912, USA — The motion of a suspension of red blood cells (RBCs) flowing in bifurcations is investigated using both low-dimensional RBC (LD-RBC) and multiscale RBC (MS-RBC) models based on dissipative particle dynamics (DPD). The blood flow is first simulated in a symmetric geometry between the diverging and converging channels to satisfy the periodic flow assumption along the flow direction. The results show that the flowrate ratio of the daughter channels and the feed hematocrit level has considerable influence on blood-plasma separation. We also propose a new method to model the inflow and outflow boundaries for the blood flow simulations: the inflow at the inlet is duplicated from a fully developed flow generated by DPD fluid with periodic boundary conditions; the outflow in two adjacent regions near the outlet is controlled by adaptive forces to keep the flowrate and velocity gradient equal, while the particles leaving the microfluidic channel at the outlet at each time step are removed from the system. The simulation results of the developing flow match analytical solutions from continuum theory. Plasma skimming and the *all-or-nothing* phenomenon of RBCs in bifurcation have been investigated in the simulations. The simulation results are consistent with previous experimental results and theoretical predictions.

[1]This work is supported by the NIH Grant R01HL094270.

3:07PM D6.00005 Identification of viscous droplets' physical properties that determine droplet behaviors in inertial microfluidics , SOOJUNG CLAIRE HUR, Rowland Institute at Harvard University — Inertial effects in microfluidic systems have recently recognized as a robust and passive way of focusing and ordering microscale particles and cells continuously. Moreover, theoretical analysis has shown that there exists a force away from channel walls in Poiseuille flow that locates deformable particles closer to the channel center than rigid counterparts. Then, the particle deformability can be extrapolated from the positions of particles with known sizes in the channel. Here, behaviors of various viscous droplets in inertial flow were investigated to identify critical properties determining their dynamic lateral position. Fluorinated oil solutions (μ =1.7mPas and 5mPas) containing droplets (1mPas$<\mu<$1.3Pas) were injected into a microfluidic channel with a syringe pump (8$<R_c<$50). Interfacial tension between aqueous and oil phases were varied by adding controlled amount of a surfactant. The diameter, a, deformability, *Def*, and dynamic lateral position, X_{eq}, were determined using high-speed microscopy. X_{eq}, was found to correlate with the particle Capillary Number, Ca_P, regardless of droplet viscosities when Ca_P <0.02 or Ca_P >0.2, suggesting that the viscous drag from the continuous phase and the interfacial tension were competing factors determining X_{eq}. Experimental results suggested that (i) interplay among droplet's viscosity, interfacial tension and inertia of carrier fluid determines dynamic lateral position of droplets and (ii) the dominant property varies at a different regime.

3:20PM D6.00006 Forces on near-wall dielectric microparticles in combined electroosmotic and Poiseuille flow through microchannels[1] , MINAMI YODA, NECMETTIN CEVHERI, Georgia Institute of Technology — Recent studies of electroosmotic (EO) flows have shown that neutrally buoyant radii a = O(0.1-1 μm) particles experience a "dielectrophoretic-like" repulsive force whose magnitude scales as a^2 [*Phys. Fluids* **18**:031702; *Langmuir* **27**:11481]. Tracers with different sizes could then have different velocities in the same nonuniform flow. Evanescent-wave particle velocimetry was therefore used to study a = 125 nm and 245 nm fluorescent polystyrene (PS) tracers in combined EO and Poiseuille flow, which is effectively the superposition of simple shear and uniform flows within 1 μm of the wall. For "coflow," where the EO and Poiseuille flows are in the same direction, the larger particles are strongly repelled from the wall; surprisingly, estimates of the magnitude of the repulsive force exceed the sum of the dielectrophoretic-like force and the shear induced electrokinetic lift force [*J Colloid Interf Sci* **175**:411]. For "counterflow," where the EO and Poiseuille flows are in opposite directions, these particles are instead *attracted* to the wall. These unexpected results suggest that the nonlinear interaction between the electric field and shear could be used to manipulate near-wall microparticles.

[1]Supported by ARO and NSF

3:33PM D6.00007 Measurement and characterization of lift forces on drops and bubbles in microchannels , CLAUDIU STAN, SLAC National Accelerator Facility, LAURA GUGLIELMINI, Center for Turbulence Research, Stanford University, AUDREY ELLERBEE, Department of Electrical Engineering, Stanford University, DANIEL CAVIEZEL, ASCOMP GmbH, Zurich, Switzerland, GEORGE WHITESIDES, Department of Chemistry and Chemical Biology, Harvard University, HOWARD STONE, Department of Mechanical and Aerospace Engineering, Princeton University — The transverse motion of drops and bubbles within liquids flowing in pipes and channels is determined by the combination of several types of hydrodynamic lift forces with external forces. In microfluidic channels, lift forces have been used to position and sort particles with high efficiency and high accuracy. We measured lift forces on drops and bubbles and discriminated between different lift mechanisms under conditions characterized by low particle capillary numbers ($0.0003 < \mathrm{Ca}_P < 0.3$) and low particle Reynolds numbers ($0.0001 < \mathrm{Re}_P < 0.1$). The measured lift forces were often much larger (up to a factor of 1000) than the predictions of analytical models of inertial and deformation-induced lift, indicating that another lift mechanism was the largest contributor to the total lift force. The systems we investigated exhibited either (i) a deformation-induced lift force enhanced by confinement effects, or (ii) a lift force for which to our best knowledge is based on physicochemical effects at the interfaces of drops and bubbles. We will present new experimental data that supports a dynamic interfacial mechanism for the second type of lift force, and discuss possible avenues for creating an analytical model for it.

3:46PM D6.00008 ABSTRACT WITHDRAWN —

3:59PM D6.00009 Viscoelasticity of dilute capsule suspension under Stokes flows , DAIKI MATSUNAGA, YOHSUKE IMAI, TAKAMI YAMAGUCHI, TAKUJI ISHIKAWA, Tohoku University — A capsule is a liquid drop enclosed by a deformable membrane. Though the capsule deformation and suspension rheology in a simple shear flow is well understood, study of those in oscillating shear flow has been limited to small deformation theory. We investigated the viscoelasticity of dilute capsule suspension by applying an oscillating shear flow. We a used numerical method developed by Walter et al., in which the boundary element method for fluid mechanics is coupled with the finite element method for membrane mechanics. Simulations were performed by changing three parameters: capillary number, viscosity ratio and non-dimensional frequency of the applied shear. We found that the maximum deformation keeps a constant value in the low frequency range, while it is inversely proportional to the frequency in the high frequency range. The result of viscoelasticity suggests that both the capillary number and viscosity ratio are important parameters in the low frequency range, while only the viscosity ratio affects the viscoelasticity in the high frequency range.

4:12PM D6.00010 Harnessing Passive Cilia for Surface Cleaning of Microfluidic Devices , ANURAG TRIPATHI, HENRY SHUM, ANNA BALAZS, University of Pittsburgh — Many biological organisms, such as mollusks and corals, utilize active cilia to prevent settlement of various fouling agents and debris on their surfaces. Inspired by these examples, we investigate if passive, non-actuated cilia can be harnessed for surface cleaning applications by utilizing oscillations in the ambient flow. By mimicking the oscillating shear flow near a ciliated wall in a channel, we show, by means of computational modeling, that the waving motion of cilia due to the oscillations in the flow can repel sticky, adhesive particles away from the surface. The results can be understood by means of a theoretical model by considering the motion of a particle penetrating an oscillating elastic layer and accounting for elastic, adhesive and hydrodynamic forces on the particle. The findings suggest a novel surface cleaning and fouling prevention mechanism for microfluidic devices dealing with transport and processing of microparticles.

Sunday, November 24, 2013 2:15PM - 4:25PM –

Session D7 Microfluids: Oscillation 329 - Sascha Hilgenfeldt, University of Illinois

2:15PM D7.00001 Probing the rheology of viscous fluids using microcantilevers and the fluctuation-dissipation theorem[1] , BRIAN ROBBINS, MILAD RADIOM, Virginia Tech, JOHN WALZ, University of Kentucky, WILLIAM DUCKER, MARK PAUL, Virginia Tech — A microscopic understanding of the rheology of fluids at high frequencies remains an important and open challenge. Current microrheology approaches include the use of micron-scale beads held in optical traps as well as micron-scale cantilevers. Typically, these approaches have been limited in their range of accessible frequencies and dynamic viscosities. In this talk we are interested in the high-frequency regime for very viscous fluids where one must include inertial effects and the frequency dependence of the viscous damping. We present experimental results of the noise spectrum in displacement of the tip of a microcantilever for a variety of fluids that cover a range of viscosities. Using analytical predictions based upon the fluctuation-dissipation theorem, we present an approach to quantify the density and viscosity of the fluid from measurements of the noise spectrum. We are particularly interested in exploring fluids much more viscous than water. We use insights from this study to explore the dynamics of an oscillating elastic object in a power-law fluid to probe the rheology of a non-Newtonian fluid at high frequency.

[1]NSF Award CBET-0959228

2:28PM D7.00002 Controlled microparticle transport in arrays of oscillating probes[1] , KWITAE CHONG, JEFF D. ELDREDGE, University of California, Los Angeles — A probe of circular cross section, undergoing rectilinear oscillation, creates large-scale steady circulatory cells by viscous streaming. In previous work, we have shown that inertial particles can be trapped inside these streaming cells, regardless of particle size and density and Reynolds number (Chong et al., Physics of Fluids, 2013). In the present work, we extend this study to various arrangements of oscillating probes. High fidelity computations are used to simulate the flow field, and a modified form of the Maxey-Riley equation is used to capture particle transport. It is shown that, by controlling the sequence of starting and stopping the oscillation of individual probes, inertial particles can be transported in a predictable manner between trapping points. In order to reduce the considerable expense of generating the flow field, we also explore the use of steady Stokes flow to serve as an approximate surrogate for the flow between probes. The boundary conditions for this flow are obtained by matching with the inner Stokes layer solution.

[1]Support for this work by the National Science Foundation, under Award Nos. CMMI-0969869 and CMMI-1000656, is gratefully acknowledged.

2:41PM D7.00003 Streaming driven by sessile microbubbles: Explaining flow patterns and frequency response , BHARGAV RALLABANDI, CHENG WANG[1], LIN GUO, SASCHA HILGENFELDT, Department of Mechanical Science and Engineering, University of Illinois at Urbana-Champaign — Ultrasound excitation of bubbles drives powerful steady streaming flows which have found widespread applications in microfluidics, where bubbles are typically of semicircular cross section and attached to walls of the device (sessile). While bubble-driven streaming in bulk fluid is well understood, this practically relevant case presents additional complexity introduced by the wall and contact lines. We develop an asymptotic theory that takes into account the presence of the wall as well as the oscillation dynamics of the bubble, providing a complete description of the streaming flow as a function only of the driving frequency, the bubble size, and the physical properties of the fluid. We show that the coupling between different bubble oscillation modes sustains the experimentally observed streaming flow vortex pattern over a broad range of frequencies, greatly exceeding the widths of individual mode resonances. Above a threshold frequency, we predict, and observe in experiment, reversal of the flow direction. Our analytical theory can be used to guide the design of microfluidic devices, both in situations where robust flow patterns insensitive to parameter changes are desired (e.g. lab-on-a-chip sorters), and in cases where intentional modulation of the flow field appearance is key (e.g. efficient mixers).

[1]Current address: Department of Mechanical and Aerospace Engineering, Missouri University of Science and Technology

2:54PM D7.00004 Calibration of the Modal Parameters of a Microcantilever from Gas Dissipation[1] , CHARLES LISSANDRELLO, KAMIL L. EKINCI, Department of Mechanical Engineering, Boston University — We determine the modal mass and the spring constant of a microcantilever from fluidic dissipation measurements. In all experiments the device is held in a vacuum chamber, and its oscillations are monitored using a sensitive heterodyne interferometer. First, thermal fluctuations of the device are measured, and the modal parameters are established. Second, the microcantilever is driven, and its dissipation is measured as a function of the gas pressure in the chamber. These dissipation measurements, combined with a theory to describe gas damping in the kinetic regime, allow us to estimate the effective modal mass and the spring constant. The measurements are repeated for multiple mechanical modes of the same device and for multiple devices. All modal parameters from the dissipation measurements are compared to those obtained from the thermal noise measurements and are found to be in excellent agreement.

[1]Support from the US NSF (through grants CMMI-0970071 and DGE-0741448) is gratefully acknowledged.

3:07PM D7.00005 Flow induced vibrations of high-frequency microcantilevers , TAEJOON KOUH, Kookmin University, Boston University, SETH HODSON, VICTOR YAKHOT, KAMIL EKINCI, Boston University — Here we present a parametric study of flow induced vibrations of high-frequency microcantilevers with resonance frequencies in the range 70 kHz to 400 kHz. In the experiments, the microcantilevers are placed in a microchannel; subsequently, a known air flow rate is established through this microchannel, while the pressure drop across is monitored. The resulting transverse vibrations of the microcantilevers are monitored optically with a displacement sensitivity at the level of thermal fluctuations as a function of the air flow rate. As the flow rate is increased, we detect a sudden increase in the vibration amplitudes of the microcantilevers. In addition, the resonance frequencies and the line-widths of the microcantilevers shift as a function of the imposed flow rate. We discuss possible trigger mechanisms for the observed vibrations, including vortex shedding and turbulent fluctuations, and obtain scaling relations in terms of the experimental parameters.

3:20PM D7.00006 Analysis of flow characteristics for viscosity sensing applications of suspended microchannel resonators , WOOK LEE, JUNGCHUL LEE, SEONGWON KANG, Dept. of Mechanical Engineering, Sogang University, Korea — In this study, we analyzed the flow characteristics and performance of a viscosity sensor based on a suspended microchannel resonator (SMR). First, we verified the assumptions of Sader et al. (2010) for their analytic solution using the approach of direction numerical simulation. Second, the relationship between monotonicity of the quality factor and the changes of integrated energy variables was investigated. It was found that the monotonicity to the Reynolds number is strongly dependent on a source term of the kinetic energy equation. Based on this, a change in the quality factor was related to specific patterns of the velocity and vorticity fields. Third, the effects of geometrical parameters of the SMR on performance as a viscosity sensor were investigated. The variations in the measurable viscosity range as well as the viscosity resolution were investigated in terms of the flow characteristics affected by the design parameters. It was found that the off-axis displacement shows a significant but consistent effect on performance of the SMR viscometer regardless of the flow condition. In contrast, the other geometric parameters show more complicated effects, as they are also related to the resonant frequency of the SMR and affected by the compressibility of a fluid.

3:33PM D7.00007 Mode Coupling of phonons in a Dense One-Dimensional Microfluidic Crystal , JEAN-BAPTISTE FLEURY, Experimental Physics, Saarland University, 66123 Saarbrücken, Germany, ULF D. SCHILLER, Institute for Complex Systems, Forschungszentrum Jülich, 52425, Germany, SHASHI THUTUPALLI, Max Planck Institute for Dynamics and Self-Organization, 37077 Göttingen, Germany, GERHARD GOMPPER, Institute for Complex Systems, Forschungszentrum Jülich, 52425, Germany, RALF SEEMANN, Max Planck Institute for Dynamics and Self-Organization, 37077 Göttingen, Germany, FLEURY/THUTUPALLI/SEEMANN TEAM, SCHILLER/GOMPPER TEAM — Microfluidic crystals are highly ordered arrangements of water-in-oil droplets flowing in microchannels. Their collective dynamics can exhibit a rich behaviour due to the long-range hydrodynamic interactions mediated by the surrounding phase. In this work, we report the specific excitation of long-lived phonon modes in a dense microfluidic crystal. The excited vibrations show transverse modes that originate from the dipolar flow field around the droplet, whereas the longitudinal modes arise from a non-linear coupling due to the breaking of translational invariance under confinement.

3:46PM D7.00008 Viscous damping of a periodic perforated microstructure , DOREL HOMENTCOVSCHI, BRUCE MURRAY, RON MILES, Mech. Eng. SUNY Binghamton — The study of a thin air layer squeezed between a moving plate and a rigid plate is important in many microelectromechanical systems such as microphones, microaccelerometers and resonators as some examples. The horizontal motion of the thin air gap in a planar microstructure yields squeeze-film damping that can adversely affect the dynamic response of the device. The backplate often contains a regular array of perforations in order to reduce the time required for fabrication. In order to analyze the viscous damping in some cases, it is possible to take advantage of the regular hole pattern by assuming periodicity. Here a method is developed to calculate the damping coefficient in microstructures with periodic perforations. An approximate analytic solution as well as numerical solutions to the incompressible Stokes equations are obtained. The results can be used to minimize squeeze film damping. In addition, since micromachined devices have finite dimensions, the periodic model for the perforated microstructure requires the calculation of some frame (edge) corrections. Analysis of the edge corrections has also been performed. Results from analytical formulas and numerical simulations match very well with published measured data.

3:59PM D7.00009 Three-dimensional tracking of acoustophoretic particle trajectories in a Poiseuille flow[1] , RUNE BARNKOB, MASSIMILIANO ROSSI, ALVARO G. MARIN, CHRISTIAN J. KÄHLER, Bundeswehr University Munich — Acoustics in microfluidics has proved as an excellent technique for particle separation. The technique is often based on advecting the particles by a Poiseuille flow, while acoustic forces push the particles transversely across the flow according to particle size, density, and compressibility. In this work we study such particle trajectories in a microchannel containing a dilute particle suspension. The microchannel is excited in its transverse ultrasound half-wave resonance, while a Poiseuille flow is imposed along the channel. In addition to the viscous drag force from the imposed flow, the particles are subject to forces from acoustic radiation as well as viscous drag from acoustic streaming (Muller et al., PRE, in press, 2013). In the microchannel cross-section, the acoustic streaming is two-dimensional, while the acoustic radiation force is one-dimensional. However, the actual particle velocity induced by the acoustic radiation force has a two-dimensional character due to wall-enhancement of the viscous Stokes drag. In the experiments, we use a 3D astigmatic particle tracking technique (APTV, Cierpka et al., Meas Sci Technol 22, 2011) to determine the particle trajectories, which we compare to theoretical predictions for future optimization of acoustic separation systems.

[1]DFG, grant no. KA 1808/16-1

4:12PM D7.00010 Shape Morphing of an Elastic Cylinder via Time-Varying Internal Viscous Flows[1] , SHAI ELBAZ, AMIR GAT, Technion - Israel Institute of Technology — Viscous flows in contact with an elastic body apply both pressure and shear stress on the solid-liquid interface and thus create internal stress- and deformation-fields within the solid structure. We study the interaction between elastic slender axi-symmetric structures and internal time-varying viscous flows as a tool to create controlled shape-morphing of such elastic cylindrical structures. We neglect inertia in the liquid and solid and focus on two cases. Case 1 is viscous flow through a hollow elastic cylinder and case 2 is axial flow in the shallow gap created by two concentric cylinders, where the internal cylinder is rigid and the external elastic. For case 1, we obtain a linear diffusion equation and for case 2 we obtain a non-linear diffusion equation governing the deformation. Solutions for both cases allowing control of the time varying deformation field by way of controlling the liquid pressure at the inlet and outlet are presented. This research is of interest to applications such as micro-swimmers and soft-robotics.

[1]This research was supported by the ISRAEL SCIENCE FOUNDATION (Grant No. 818/13)

Sunday, November 24, 2013 2:15PM - 4:12PM –

Session D8 Particle-Laden Flows II: Experimental Studies 330 - Franklin Shaffer, Albany Research Center

2:15PM D8.00001 DNS of Oscillatory Boundary Layer Over a Closely Packed Layer of Sediment Particles[1]

, CHAITANYA D. GHODKE, JOSEPH SKITKA, SOURABH V. APTE, Oregon State University — Fully resolved direct numerical simulations are performed using fictitious domain approach (Apte et al., JCP 2009) to investigate oscillatory turbulent flow over a rough wall corresponding to the experimental work of Keiller & Sleath (JFM 1976) and representative of a sediment layer in coastal environments. Four particle Reynolds numbers in the range, $Re = 660 - 2780$ are studied for a fixed sphere size and results are compared against available experimental data. Flow is characterized in terms of coherent vortex structures, Reynolds stress variation, turbulent cross-correlations and PDF distributions. Reynolds stress is negative during the first half of the cycle as ejection and sweep events prevail and is positive during the second half where outward/inward interactions of motion are dominant. The net lift coefficient remains positive over the cycle with a period which is half of that of the forcing function and is well correlated with phase averaged velocity square (U^2). The pressure contribution towards the net lift force is found to be more dominant over the viscous contribution. Also the PDF distributions of velocity fluctuations show non-Gaussian behavior. These detailed findings are useful in improving the criterion for predicting sediment incipient motion.

[1]NSF project #1133363, Sediment-Bed-Turbulence Coupling in Oscillatory Flows

2:28PM D8.00002 4-Frame Particle Tracking Based on PIV to Study Inertial Particle Relative Motion in Isotropic Turbulence[1]

, LUJIE CAO, Department of Mechatronic, Ocean University of China, Qingdao 266003, China, ZHONGWANG DOU, ZACHARY PECENAK, FAN YANG, ZACH LIANG, HUI MENG, Department of Mechanical & Aerospace Engineering, State University of New York at Buffalo, Buffalo, NY 14260 — The statistics of particle pair relative velocity is a critical parameter in particle collision models. Experimental measurements are required to quantify the dependence of inertial particle relative velocity on turbulence and particle parameters. Our aim is to develop a technique that can directly measure the dynamics of inertial particles relative velocity statistics, in high Re number, lab- generated isotropic turbulence. The technique is based on 4-frame particle tracking using two sets of regular PIV systems, each recording a pair of correlated particle images, with additional synchronization controlling the temporal relationship between two PIV sets. To ensure spatial correlation between coordinates in images, we (1) use perpendicularly polarized laser illumination to label particle scattering with corresponding PIV recording, (2) make the two illumination laser sheets overlap at the test field, (3) arrange two PIV cameras orthogonally after a polarizing beam splitter to record the particle field. This allows us to capture 4 consecutive frames of particle images with a certain spatial shift. PTV algorithm is applied to further extract particle trajectories, from which we directly measure particle position and absolute velocity and further deduce particle relative velocity.

[1]This research is funded by an NSF grant CBET-0967407.

2:41PM D8.00003 Simultaneous measurements of velocity gradients and rod rotation in 3D turbulence[1]

, STEFAN KRAMEL, RUI NI, GREG VOTH, Wesleyan University, NICHOLAS OUELLETTE, Yale University — When anisotropic particles are advected in a fluid flow, they rotate in response to the velocity gradient tensor. In 3D turbulent flows, it has previously not been possible to experimentally measure both the motion of anisotropic particles and the velocity gradients simultaneously. We have built a scanning particle tracking velocimetry system in which we illuminate a narrow slab of the volume of interest and scan the illuminated slab through the entire volume, taking sequential images with four high speed cameras. Compared to full volume illumination, this technique enables us to greatly increase the particle concentration because it removes many stereo-matching ambiguities, resulting in a high spatial resolution of the fluid velocity. The trade-off is that the temporal resolution is decreased. We image a low concentration of rods in addition to a high concentration of tracer particles in order to allow extraction of the velocity gradient tensor at the positions of the rods. Rods are found to preferentially align with the direction of the vorticity vector and the intermediate strain-rate eigenvector.

[1]Support from NSF grant DMR-1208990

2:54PM D8.00004 ABSTRACT WITHDRAWN –

3:07PM D8.00005 Interfacial deflection and jetting of a paramagnetic particle-laden fluid

, IAN GRIFFITHS, Mathematical Institute, University of Oxford, SCOTT TSAI, Ryerson University, ZHENZHEN LI, ESPCI, PILNAM KIM, KAIST, HOWARD STONE, Princeton University — Arsenic removal from contaminated water is a formidable challenge in the water-purification industry. A promising technique to remove such contaminants utilizes magnetic separation, whereby arsenic adsorbs onto magnetic nanoparticles that are dispersed in the contaminated water. These magnetically tagged clusters are then collected safely using a magnetic field gradient that pulls the aggregates to the liquid–air interface, which induces an interfacial deflection before transitioning into a jet of magnetic material. We present experiments that are motivated by the applications of collecting aggregates of magnetic material, in which colloidal magnetic particles are first mixed with water, then collected at a liquid-air interface by an applied magnetic field from a permanent magnet. We derive a mathematical model that predicts the interfacial deflection and ejection of magnetic material, and compare the results with the experimental observations. The model is reduced by exploiting the small interfacial deflection for low magnetic fields and is used to make predictions that are more difficult to glean from the experimental observations alone. We draw conclusions on the model's ability to offer a route to design optimization for water purification strategies.

3:20PM D8.00006 An experimental investigation of the settling and resuspension of gravity-driven, mono- and bi-disperse slurries

, GILBERTO URDANETA, UCLA, MATT HIN, Cornell University, KAIWEN HUANG, UCLA, SHREYAS KUMAR, Harvey Mudd College, ALIKI MAVROMOUSTAKI, JEFFREY WONG, UCLA, SUNGYON LEE, Texas A&M, ANDREA BERTOZZI, UCLA — We investigate the dynamics of gravity-driven mono- and bidisperse suspensions consisting of silicone oil and negatively buoyant particles of different densities experimentally. The well-mixed slurry mixtures spread down an inclined plane, exhibiting distinct flow patterns arising from competition of gravitational sedimentation and varying shear forces. We confirm the results of previous studies where, an initially well-mixed flow evolves towards either a "settled" regime in which the particles settle to the substrate, or a "ridged" regime in which the particles aggregate at the front of the flow. Our results show that the addition of a second particlespecies induces or prevents the setting of particles due to the mismatch in particle densities. We show that the latter depends strongly on the relativeamountof heavy to light particles used. We compare our experimental results to spreading relations as applied to particle-free, thin-fluid films. Further,we investigate the evolution of particle concentrations in each of the two regimes by using fluorescent particle beads and compare our results toexistingtheoretical models.

3:33PM D8.00007 Particle-turbulence interaction of suspended load by forced jet impinging jet on a mobile sediment bed[1] , RAHUL MULINTI, KYLE CORFMAN, KEN KIGER, University of Maryland — Particle suspension and transport induced by a forced impinging jet on a sediment layer has been investigated experimentally using two-phase particle image velocimetry (PIV). Glass spheres of two size classes, 45-63 micrometers and 120-180 micrometers have been used, with focus on transport characteristics during both the early development time (when the bed is relatively flat and smooth just following the passage of starting transients) and at late times (when a significant erosion bedforms are present in the form of annular wave-like patterns). Preferential mobilization and suspension has been observed at the crests of these waves and deposition due to entrainment of particles by the secondary, counter-rotating vortex formed by boundary layer separation near the surface. 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 also been studied. The effect of the changing sediment bed profile on sediment erosion rates has also been examined briefly.

[1]This work is supported by the AFSOR under grant FA95500810406.

3:46PM D8.00008 Laboratory Study of Homogeneous and Isotropic Turbulence at High Reynolds Number[1] , ZACHARY PECENAK, ZHONGWANG DOU, FAN YANG, Department of Mechanical & Aerospace Engineering, State University of New York at Buffalo, Buffalo, NY 14260, LUJIE CAO, Department of Mechatronic, Ocean University of China, Qingdao 266003, China, ZACH LIANG, HUI MENG, Department of Mechanical & Aerospace Engineering, State University of New York at Buffalo, Buffalo, NY 14260 — To study particle dynamics modified by isotropic turbulence at high Reynolds numbers and provide experimental data for DNS validation, we have developed a soccer-ball-shaped truncated icosahedron turbulence chamber with 20 adjoining hexagon surfaces, 12 pentagon surfaces and twenty symettrically displaced fans, which form an enclosed chamber of 1m diameter. We use Particle Image Velocimetry (PIV) technique to characterize the base turbulent flow, using different PIV set ups to capture various characteristic scales of turbulence. Results show that the stationary isotropic turbulence field is a spherical domain with diameter of 40 mm with quasi-zero mean velocities. The maximum rms velocity is $\sim$1.5 m/s, corresponding to a Taylor microscale Re of 450. We extract from the PIV velocity field the whole set of turbulent flow parameters including: turbulent kinetic energy, turbulent intensity, kinetic energy dissipation rate, large eddy length and time scales, the Kolmogorov length, time and velocity scales, Taylor microscale and Re, which are critical to the study of inter-particle statistics modified by turbulence.

[1]This research is funded by an NSF grant CBET-0967407.

3:59PM D8.00009 Bead resuspension and saltation in a turbulent channel flow , RENE VAN HOUT, Technion-Israel Institute of Technology — Resuspension and saltation of near neutrally buoyant, polystyrene beads in a turbulent boundary layer was studied using TR-PIV and PTV in a horizontal, water channel facility (Re = 7353). Near wall coherent structures were visualized using spatial distributions of vorticity and swirling strength in combination with instantaneous u_1u_2 correlations and u_1. Two case studies, (i) on resuspension and (ii) on saltation showed that lift-off coincided with vortex core passage creating an ejection-sweep cycle. In all cases, beads left the wall when immersed in near-wall ejections and exposed to positive shear. A high shear induced lift force coincided with bead lift-off while the Magnus force and translation induced lift were negligible. The wall-normal component of the drag force mostly opposed lift-off. The difference between resuspension and saltation was governed by the type of coherent structures the beads encountered when lifted out of the viscous sublayer. Resuspension occurred when beads were carried upwards by a combination of a strong, spatially coherent upstream fast moving flow structure and a downstream ejection. On the other hand, saltation was accompanied by similar albeit weaker and spatially less coherent near-wall turbulence structures.

Sunday, November 24, 2013 2:15PM - 4:25PM –

Session D9 Instability: Interfacial and Thin-Film II 333 - Demetrios Papageorgiou, Imperial College London

2:15PM D9.00001 Influence of surfactant concentration on satellite formation from the rupture of viscous liquid filaments[1] , EMILIA NOWAK, MARK SIMMONS, University of Birmingham, RICHARD CRASTER, OMAR MATAR, Imperial College London — Drop formation from the rupture of liquid filaments are critically affected by the presence of liquid soluble surfactants, particularly above the critical micelle concentration (CMC). In this paper, we apply the long wave approximation to elucidate the interfacial topology leading to the formation of droplets and satellites from the rupture of a cylindrical fluid filament. The numerical results are compared with experiments performed using aqueous filaments in a continuous phase of silicone oil, using an aqueous phase soluble surfactant (SLES) over a range of concentrations above and below the CMC. Comparisons are made as a function of the capillary number and viscosity of the continuous phase, focusing on the temporal variation in minimum filament radius, contact angle and the number and size of the droplets formed. Similarities and differences between the experiments and the model are noted.

[1]EPSRC Programme Grant EP/K003976/1

2:28PM D9.00002 Surfactant-driven dynamics of immiscible jets under microfluidic confinement[1] , JOAO CABRAL, JUNFENG YANG, OMAR MATAR, Imperial College London — We examine the dynamics of three water jets in oil (PDMS) under microfluidic confinement in the presence of surfactant (sodium dodecyl sulphate). Our experimental results demonstrate the occurrence of two flow regimes, "jetting" and "dripping," depending on the choice of system parameters; the latter are the flow-rates of water and oil, the viscosity ratio, and the surfactant concentration. In the dripping regime, the average diameter of the water droplets decreases with increasing oil flow-rate until a transition to jetting occurs. In the jetting regime, and at high oil and water flow-rates, and high oil viscosity, our results demonstrate that each jet exhibit sinusoidal deformations that appear to be either in- or out-of-phase with those of their neighbours'. Numerical simulations of the system studied experimentally are also carried out using a volume-of-fluid approach, which account for the presence of insoluble surfactant. The results of these simulations capture the trends observed in the experiments.

[1]Skolkovo Foundation, UNIHEAT project

2:41PM D9.00003 The influence of evaporation on instabilities of liquid layer with insoluble surfactant , ALEXANDER MIKISHEV, Sam Houston State University, ALEXANDER NEPOMNYASHCHY, Techion-IIT — A horizontally infinite layer of an evaporating incompressible Newtonian liquid with insoluble surfactant on the free deformable surface is studied theoretically. The layer is subjected to a transverse temperature gradient. The evaporation process is described by 2D one-side model based on the assumptions of density, viscosity and thermal conductivity of the gaseous phase being small compared to the same properties of the liquid phase. Surface tension of the liquid-vapor surface linearly depends on temperature and concentration of surfactant. On the basis of experiments we assume that thermal resistance to the evaporation at the interface is a linear function of surfactant concentration. The evaporation mass flux depends on the interface temperature and vapor pressure. Using the long-wave approximation and assumption of slow time evolution the system of nonlinear equations is obtained. The equations retain all relevant physical effects which take place in the system. Linear stability analysis of the base state in the case of non-equilibrium evaporation is performed. The results are compared to those of the non-evaporating case.

2:54PM D9.00004 The stabilizing mechanism of surfactants in falling films[1] , VASILIS BONTOZOGLOU, GEORGE KARAPETSAS, University of Thessaly — We investigate the stabilizing effect of surfactants in gravity-driven liquid films flowing down inclined surfaces. To this end, we consider the Navier-Stokes equations along with advection-diffusion equations and chemical kinetic fluxes for the surfactant transport and derive an analytical solution by expanding in the limit of long-wave disturbances. We present a physical mechanism for the role of a surfactant of arbitrary solubility. The stabilizing effect is due to an interfacial concentration gradient which is in-phase with the interfacial deformation inducing Marangoni stresses driving liquid from the crest to the trough. The strength of the interfacial concentration gradient is shown to be maximum for an insoluble surfactant and to decrease with increasing surfactant solubility. The decrease is explained in terms of the phase of mass transfer between interface and bulk, which mitigates the interfacial flux by the flow perturbation, and leads to the attenuation of Marangoni stresses.

[1]The authors would like to acknowledge the financial support by the General Secretariat of Research and Technology of Greece under the Action "Supporting Postdoctoral Researchers" (grant number: PE8/906)

3:07PM D9.00005 Nonlinear phenomena in two-fluid shear flows in the presence of surfactants , ANNA KALOGIROU, DEMETRIOS PAPAGEORGIOU, Imperial College London — The flow of two superposed fluids in a channel in the presence of an insoluble surfactant is studied. The surfactant is present at the interface in a dilute concentration. Asymptotic analysis in the limit of a thin lower layer is performed and a system of coupled weakly nonlinear evolution equations is derived. The system describes the evolution of the film thickness and the local surfactant concentration. A novel feature is the presence of a nonlocal term due to multiphase coupling. The system of nonlinear evolution equations is solved numerically and the effect of surfactants on the dynamics is investigated. Numerical experiments for zero and for finite Reynolds numbers indicate that the solutions are mostly nonlinear travelling waves of permanent form or time-periodic travelling waves. As the length of the system increases, the dynamics of the system become more complex and include quasi-periodic and chaotic solutions.

3:20PM D9.00006 Numerical Study of a Hydrodynamic Instability Driven by Evaporation[1] , SERGIO HERNANDEZ-ZAPATA, JULIO CESAR RUBEN ROMO-CRUZ, ERICK JAVIER LOPEZ-SANCHEZ, GERARDO RUIZ-CHAVARRIA, Facultad de Ciencias, UNAM — The study of hydrodynamic instabilities in liquid layers produced by evaporation has several applications on industry and technology. In this work we study numerically the conditions under which a liquid layer becomes unstable when evaporation in the vapor-liquid interphase is present. The evaporation process follows the Hertz-Knudsen law (the evaporation rate is proportional to the difference between the saturated vapor pressure at the liquid layer temperature and the vapor partial pressure in the environment). Additionally to the usual boundary conditions on solid walls (for example, the non-slip condition for the velocity), we analyze the boundary conditions in the vapor-liquid interphase where the momentum and energy balances have to be taken into account and where the evaporation plays a crucial role. To solve this problem the linear theory of stability is used; that is, a small perturbation around the basic solution is applied (flow at rest and a temperature stationary field). The equations are solved using the Chebyshev pseudo-spectral method. The results are compared with the more usual Rayleigh-Bénard and Marangoni mechanisms as well as with some experiments carried out by our team.

[1]Authors acknowledge DGAPA-UNAM by support under project IN116312, "Vorticidad y Ondas no lineales en fluidos."

3:33PM D9.00007 Long-wave Marangoni convection in a heated liquid layer with insoluble surfactant[1] , MATVEY MOROZOV, ALEX ORON, ALEXANDER NEPOMNYASHCHY, Technion - Israel Institute of Technology — Recently, long-wave Marangoni convection in a heated binary-liquid layer was considered by Podolny *et al.* (Phys. Fluids **18**, 054104, 2006) revealing rich dynamics stemming from oscillatory instability. These results were obtained in the absence of surfactants. In the present work we investigate an opposite limit: a liquid layer with insoluble surfactant. We consider a liquid layer lying on a solid horizontal substrate with insoluble surfactant adsorbed at the deformable free surface. Convection is triggered by a given transverse temperature gradient. Long-wave linear stability analysis of the quiescent state of the layer reveals a competition between monotonic and oscillatory modes of instability. We derive nonlinear evolution equations governing the large-scale dynamics of the layer. Linear stability analysis of these equations indicates their applicability only in the case of oscillatory instability. We then carry out weakly nonlinear analysis in the vicinity of the oscillatory-instability threshold for the case of a 2D layer, and study corresponding pattern selection. Finally, we compare the analytical results with the numerical solutions of our nonlinear evolution equations.

[1]This work is supported by the European Union via FP7 Marie Curie scheme Grant PITN-GA-2008-214919 (MULTIFLOW).

3:46PM D9.00008 Completely Stabilizing the Interface in a Rayleigh-Taylor Problem by Heating[1] , RANGA NARAYANAN, LEWIS JOHNS, University of Florida — The interface in a Rayleigh-Taylor problem can be stabilized to perturbations of any wavelength by merely heating. We present a simple formula for estimating the temperature difference required to do this. We assume that the fluid resides in a porous medium so as to simplify the algebraic manipulations and to avoid surface tension gradients.

[1]Support from NSF 0968313 is gratefully acknowledged

3:59PM D9.00009 Nonlinear dynamics of a binary liquid layer heated from above[1] , ALEXANDER NEPOMNYASHCHY, Technion – Israel Institute of Technology, Haifa, Israel, SERGEY SHKLYAEV, Institute of Continuous Media Mechanics, Ural Branch of Russian Academy of the Sciences, Perm, Russia — It is well known [Pearson, JFM, 1958] that for the Marangoni convection the critical wavenumber k_c scales as $B^{1/4}$ as the Biot number B characterizing the heat flux from the free surface tends to zero. In a layer of binary mixture [Podolny et al., Phys. Fluids, 2005], for heating from above another longwave mode, with $k_c = O(\sqrt{B})$, is important. In this work we study the nonlinear evolution of the latter mode. It is shown that the amplitude of steady convection is governed by a solvability condition for a certain linear nonhomogeneous problem. This makes possible an analytical study of finite-amplitude regimes of convection, with perturbations of the temperature and solute concentration of order unity. It is shown that up-hexagons and squares are selected on hexagonal and square lattices, respectively. On the superlattice combining both square and hexagonal lattices multistability takes place: at the Marangoni number larger than a certain critical value both squares and up-hexagons are stable.

[1]S.S. is supported within RFBR–Ural grant N13-01-96010a.

4:12PM D9.00010 Vibration impact on Marangoni instability in a thin film[1] , SERGEY SHKLYAEV, ALEXEY ALABUZHEV, Institute of Continuous Media Mechanics, Ural Branch of the Russian Academy of Sciences, Perm, Russia, MIKHAIL KHENNER, Department of Mathematics, Western Kentucky University, Bowling Green, Kentucky, USA — We study the influence of a vertical vibration on Marangoni instability in a thin film heated from below. Using a multi-scale expansion the film dynamics is considered in a wide range of the vibration frequency ω: from $\omega t_v \gg 1$ to $\omega t_g = O(1)$, where t_v is the time of viscous relaxation across the layer and t_g is the typical time of the longwave surface dynamics. We have shown that for $\omega t_g \gg 1$ there is no interaction between the Faraday instability and the Marangoni convection because of the large differences in the characteristic time- and length scales (see also [Thiele et al. , JFM (2006)]). Therefore, the averaging technique is applied to derive the equation governing the film dynamics in slow time (in comparison with $1/\omega$). We show that the vibration suppresses the Marangoni instability in a confined cavity; however, the branching remains subcritical. This amplitude equation becomes invalid for the ultra-low frequency, $\omega t_g = O(1)$. In this case the standard amplitude equation [Oron et al., Rev. Mod. Phys. (1997)] is obtained, but with the modulated gravity. The vibration does not change the stability threshold; the subcritical excitation leads to the emergence of a limit cycle instead of a film rupture.

[1]S.S. is supported by RFBR within the grant 13-01-96010a, A.A. is supported by RFBR within the grant 12-01-31366

Sunday, November 24, 2013 2:15PM - 3:59PM –

Session D10 Geophysical: Oceanographic II 334 - Sutanu Sarkar, University of California, San Diego

2:15PM D10.00001 Dynamics of SQG Vortices and Passive Scalar Transport , CECILY KEPPEL, STEFAN LLEWELLYN SMITH, MAE UCSD — The surface quasi-geostrophic (SQG) equations are a model for low-Rossby number geophysical flows in which the dynamics are governed by potential temperature dynamics on the boundary. We examine the dynamics of SQG vortices and the resulting flow in the entire fluid including at first order in Rossby number ($O(Ro)$). This requires solving an extension to the usual QG equation to compute the velocity corrections, and we demonstrate this mathematical procedure. As we show, it is simple to obtain the vertical velocity, but difficult to find the $O(Ro)$ horizontal corrections. We then consider specific cases of interactions of vortices and examine the tracer transport properties in the interior of the fluid. We show various diagnostics for examining the effect of the vertical transport.

2:28PM D10.00002 An efficient coarse grid projection method for quasigeostrophic models of large-scale ocean circulation[1] , ANNE STAPLES, OMER SAN, Virginia Tech — We present a coarse grid projection (CGP) multiscale method to accelerate computations of quasigeostrophic (QG) models for large scale ocean circulation. These models require solving an elliptic sub-problem at each time step, which takes the bulk of the computational time. The method we propose here is a modular approach that facilitates data transfer with simple interpolations and uses black-box solvers for solving the elliptic sub-problem and potential vorticity equations in the QG flow solvers. After solving the elliptic sub-problem on a coarsened grid, an interpolation scheme is used to obtain the fine data for subsequent time stepping on the full grid. The potential vorticity field is then updated on the fine grid with savings in computational time due to the reduced number of grid points for the elliptic solver. The method is applied to both single layer barotropic and two-layer stratified QG ocean models for mid-latitude oceanic basins in the beta plane. The method is found to accelerate these computations (at a linear rate) while retaining the same level of accuracy in the fine-resolution field. In addition, numerical oscillations due to lower grid resolutions are effectively eliminated with CGP method.

[1]This material is based upon work supported by the Institute for Critical Technology and Applied Science at Virginia Tech under Grant No. 118709

2:41PM D10.00003 Nonlinear Scale Interactions and Energy Pathways in the Ocean , HUSSEIN ALUIE, MATTHEW HECHT, Los Alamos National Laboratory, GEOFFREY VALLIS, Princeton University and University of Exeter — Large-scale currents and eddies pervade the ocean and play a prime role in the general circulation and climate. The coupling between scales ranging from $O(10^4)$ km down to $O(1)$ mm presents a major difficulty in understanding, modeling, and predicting oceanic circulation and mixing, where the energy budget is uncertain within a factor possibly as large as ten. Identifying the energy sources and sinks at various scales can reduce such uncertainty and yield insight into new parameterizations. To this end, we refine a novel coarse-graining framework to directly analyze the coupling between scales. The approach is very general, allows for probing the dynamics simultaneously in scale and in space, and is not restricted by usual assumptions of homogeneity or isotropy. We apply these tools to study the energy pathways from high-resolution ocean simulations using LANL's Parallel Ocean Program. We examine the extent to which the traditional paradigm for such pathways is valid at various locations such as in western boundary currents, near the equator, and in the deep ocean. We investigate the contribution of various nonlinear mechanisms to the transfer of energy across scales such as baroclinic and barotropic instabilities.

2:54PM D10.00004 Exploring the dynamics of turbulence suppression due to dispersed phase in various geophysical flows[1] , MRUGESH SHRINGARPURE, Mechanical and Aerospace Engineering, University of Florida, USA, MARIANO CANTERO, Institute Balseiro, Bariloche Atomic Center, San Carlos de Bariloche, Argentina, TIAN-JIAN HSU, Civil and Environmental Engineering, University of Delaware, USA, BALACHANDAR S., Mechanical and Aerospace Engineering, University of Florida, USA — Geophysical flows that are characterized as multiphase and turbulent, the feedback of dispersed phase tends to alter the carrier phase turbulence and degree of isotropy. In turbidity currents, suspended sediments sustain turbulence by driving the flow and at the same time can kill the flow by suppressing turbulence through stratification effects. Similarly in hurricanes, the intensity of wind can be modulated by the stratification effects of water droplets that are injected into it by sea-sprays. To study such flows, we implemented models with dilute suspensions and performed DNS at moderate Reynolds numbers. These studies show that there are three governing parameters: Reynolds number, Richardson number and size of the dispersed phase particles. Parametric groupings that quantify turbulence suppression were identified for these flows. We have also looked into the interaction of wave induced turbulence and its ability to transport fine sediments. Here we will present the mechanism that is responsible for modulating the near wall vortical structures that can potentially explain the loss of turbulence in turbidity currents, fluctuations in the carry capacity of wave induced sediment transport and drag saturation of hurricane intensity winds due to sea-sprays.

[1]We acknowledge support from US NSF through grants OCE 1131016 and OISE 0968313

3:07PM D10.00005 LES of full-depth Langmuir circulation in a crosswind tidal current , ANDRES TEJADA-MARTINEZ, NITYANAND SINHA, University of South Florida, CHESTER GROSCH, GUILLAUME MARTINAT, Old Dominion University — We report on the impact of a crosswind tidal current on full-depth Langmuir circulation (LC) in shallow water computed via large-eddy simulations (LES). 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 current with the Stokes drift velocity induced by surface gravity waves. During times of weak tidal current, full-depth LC disrupts the classical log-layer dynamics occurring at the bottom of the water column. 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 $90 < z+ < 200$. However, during times of strong tidal current, bottom-generated turbulence induced by the tide is able to break-up the full-depth LC giving rise to smaller scale LC characterized by different turbulent structure. The LC turbulent structure during strong and weak tidal currents is consistent with field measurements during episodes of full-depth LC. Statistics of the turbulence associated with LC during strong and weak tides will be contrasted.

3:20PM D10.00006 A K-profile parameterization of Langmuir turbulence in shallow water , NITYANAND SINHA, ANDRES E. TEJADA-MARTINEZ, University of South Florida, CHESTER E. GROSCH, GUILLAUME MARTINAT, Old Dominion University — Langmuir turbulence in shallow water is often characterized by full-depth Langmuir circulation (lc) generated by the interaction between the wind-driven shear current and the stokes drift velocity induced by surface gravity waves. Large-eddy simulations (LES) of full-depth LC in a wind-driven shear current have revealed that mixing due to LC erodes the bottom log-law velocity profile inducing a profile resembling a wake law. Meanwhile, near the surface, stokes drift shear serves intensify small scale eddies leading enhanced mixing and disruption of the surface log-law. A k-profile parameterization (KPP) comprised of local and nonlocal components is introduced capturing these basic mechanisms by which full-depth LC and associated turbulence impact the mean flow. Single water column Reynolds-averaged Navier-Stokes (RANS) simulations with the new parameterization are presented showing good agreement with les in terms of mean velocity profiles.

3:33PM D10.00007 Numerical Simulations of the Reduced Craik–Leibovich Equations in Spatially-Extended Domains , ZHEXUAN ZHANG, GREGORY CHINI, University of New Hampshire, KEITH JULIEN, University of Colorado, Boulder — Large-eddy simulations of the Craik–Leibovich (CL) equations, a surface-wave filtered version of the Navier–Stokes equations, have been used extensively over the last decade to investigate Langmuir turbulence in the ocean surface boundary layer (BL). However, the simulations are generally restricted to moderate-sized domains, several hundreds of meters in lateral scale; in contrast, spatially-extended arrays of quasi-coherent vortical structures in Langmuir turbulence routinely span lateral scales from 1–10 km. To facilitate simulations of Langmuir turbulence in spatially-extended domains, Chini et al. (GAFD 2009) derived an asymptotically reduced model that consistently filters streamwise variability on scales comparable to the BL depth or less and temporal fluctuations associated with rapid-distortion transients. Preliminary simulations were performed in a moderate-sized domain (fitting only 1 or 2 pairs of vortical flow structures) to verify that the model is well posed and retains the essential dynamics of Langmuir circulation. Here, a more comprehensive set of simulations in a spatially-extended domain is performed to investigate the physics and the computational efficiency of the reduced model.

3:46PM D10.00008 The Craik-Leibovich Vortex Force as a Skin Effect[1] , ZIEMOWIT MALECHA, GREGORY CHINI, University of New Hampshire, Program in Integrated Applied Mathematics, Durham, KEITH JULIEN, University of Colorado, Department of Applied Mathematics, Boulder — The CraikLeibovich (CL) equations are a surface-wave filtered version of the instantaneous NavierStokes equations in which the rectified effects of the surface waves are captured through a so-called "vortex force" term:the cross-product of the Stokes, or Lagrangian, mass drift associated with the filtered surface waves and the filtered vorticity vector.For locally generated wind waves, the Stokes drift is very strongly surface confined.In this scenario, the induced body force may be represented as a surface, or skin, effect. Using matched asymptotic analysis in this limit, we derive effective boundary conditions (BCs) for the flow beneath the Stokes drift layer (i.e. in the bulk of the mixed layer). We establish the regime of validity of the resulting formulation by performing linear stability analyses and numerical simulations of both the asymptotic model and the full CL equations for a variety of vertical Stokes drift profiles.The effective BC formulation offers both theoretical and computational advantages, and should be particularly useful for LES of Langmuir turbulence for which the need to resolve very small scale near-surface flow structures imposes severe computational constraints.

[1]GPC would like to acknowledge funding from the NSF award 0934827, administered by the Physical Oceanography Program.

Sunday, November 24, 2013 2:15PM - 4:25PM –

Session D11 Bubbles II: Cavitation, Acoustics and Biomedical 335 - Steven L. Ceccio, University of Michigan

2:15PM D11.00001 Original flocculation technique via acoustic cavitation bubbles driven by 20.3-kHz ultrasound in water , YUKI MIZUSHIMA, Graduate School of Science and Technology, Shizuoka University, TAKAYUKI SAITO, Research Institute of Green Science and Technology, Shizuoka University — Strange flocculation mechanism of particles (up to 1.0mm) driven by the acoustic field (20.3-kHz) is observed in water. It is not well-known particle formation in acoustic field, like *dust striation*, but spherical agglomeration. Because kHz-order ultrasound is not acceptable for the separation technique due to its weak-directionality, applicable particle sizes are limited as similar size to a wavelength of the irradiated ultrasound or smaller than that. Hence, particles which are larger than mm-order in diameter are difficult to be manipulated with MHz-band ultrasound. However, our flocculation technique overcomes the limitation. It deeply relates to the motion of cavitation bubbles around the particles. First, in this study, we captured the particle motion and acoustic-cavitation-oriented bubble motion simultaneously by using a high-speed video camera. Second, we measured the distribution of the sound pressure in the water phase and discussed the relationship between that of the sound pressure and the motion of the particle and the acoustic cavitation bubble. Finally, we investigated the effects of the gravity force, the acoustic radiation force and the spatial heterogeneity of the pressure acting on the particle.

2:28PM D11.00002 On the behavior of a bubble cloud under an ultrasound field[1] , ANA MEDINA-PALOMO, ELENA IGUALADA-VILLODRE, JAVIER RODRIGUEZ-RODRIGUEZ, Universidad Carlos III de Madrid — We present our latest numerical results on the determination of the resonance frequency of a bubble cloud excited by an acoustic wave. Thermal effects are incorporated to the Keller-Miksis equation by integration of an ode which models the heat transfer between the bubble and the liquid. It is found that thermal effects make the bubble oscillations damp out faster, which difficultates the resonance detection. We study how the parameters of the population, i.e. the mean and variance of the size distribution, affect the spectra, and thus, the detection of the resonance frequency. Spectra of monodisperse populations exhibit a peak at the resonance frequency that is easier distinguished compared to the case of polydisperse populations. To overcome the issue that the resonance peak is usually smaller than the central peak corresponding to the insonating wave, we focus on two strategies. Firstly, the use of a chirp, i.e. an acoustic pulse variable in frequency. This signal has a flat spectrum in a wide frequency band and is therefore more appropiate to excitate a population with different sizes. A second strategy consists in insonating the bubbles with a shock pressure wave. In the laboratory this is achieved by placing them in an open bottle that is suddenly hit at its mouth.

[1]This work has been supported by Spanish Ministries of Science and of Economy and Competitiveness through grants: DPI2008-06369 and DPI2011-28356-C03-02.

2:41PM D11.00003 Three-dimensional features on oscillating microbubbles streaming flows , MASSIMILIANO ROSSI, ALVARO G. MARIN, Bundeswehr University Munich, CHENG WANG, SASCHA HILGENFELDT, University of Illinois at Urbana-Champaign, CHRISTIAN J. KÄHLER, Bundeswehr University Munich — Ultrasound-driven oscillating micro-bubbles have been used as active actuators in microfluidic devices to perform manifold tasks such as mixing, sorting and manipulation of microparticles. A common configuration consists in side-bubbles, created by trapping air pockets in blind channels perpendicular to the main channel direction. This configuration results in bubbles with a semi-cylindrical shape that creates a streaming flow generally considered quasi two-dimensional. However, recent experiments performed with three-dimensional velocimetry methods have shown how microparticles can present significant three-dimensional trajectories, especially in regions close to the bubble interface. Several reasons will be discussed such as boundary effects of the bottom/top wall, deformation of the bubble interface leading to more complex vibrational modes, or bubble-particle interactions. In the present investigation, precise measurements of particle trajectories close to the bubble interface will be performed by means of 3D Astigmatic Particle Tracking Velocimetry. The results will allow us to characterize quantitatively the three-dimensional features of the streaming flow and to estimate its implications in practical applications as particle trapping, sorting or mixing.

2:54PM D11.00004 Shear Stress induced Stretching of Red Blood Cells by Oscillating Bubbles within a Narrow Gap , FENFANG LI, MILAD MOHAMMADZADEH, CLAUS-DIETER OHL, Division of Physics and Applied Physics, School of Physical and Mathematical Sciences, Nanyang Technological University, Singapore 637371, CLAUS-DIETER OHL TEAM — The flow pattern, especially the boundary layer caused by the expanding/contracting bubble in a narrow gap (15 μm) and the resultant stretching of red blood cells is investigated in this work. High speed recordings show that a red blood cell (biconcave shape, thickness of 1-2 μm) can be elongated to five times its original length by a laser-induced cavitation bubble within the narrow gap. However, flexible cancer cells in suspension (RKO, spherical shape, diameter of 10-15 μm) are hardly elongated under the same experimental condition. We hypothesize that the shear stress at the boundary layer is crucial for this elongation to occur. Therefore, in order to resolve the related fluid dynamics, we conducted numerical simulations using the finite element method (Fluent). The rapidly expanding/contracting vapor bubble is successfully modeled by employing viscosity and surface tension. The transient pressure inside the bubble and the velocity profile of the flow is obtained. We observe strong shear near the upper and lower boundary during the bubble oscillation. The flow fields are compared with analytical solutions to transient and pulsating flows in 2D. In the experiment the red blood cells sit within the lower boundary layer, thus are probably elongated by this strong shear flow. In contrast, the spherical cancer cells are of comparable size to the gap height so that they are lesser affected by this boundary layer flow.

3:07PM D11.00005 Using ultrasound to steer ultrasound contrast agents: Implications for targeted drug delivery , ALICIA CLARK, ALBERTO ALISEDA, University of Washington — Ultrasound can be used to manipulate ultrasound contrast agents (UCAs), micron-sized bubbles used in ultrasound imaging to increase image contrast. The Bjerknes force, resulting from the lagged response of the microbubbles to the oscillatory ultrasound pressure field, can be utilized to steer the microbubbles to a targeted area in the vasculature, with the microbubbles serving as drug delivery vectors and injectors. The response of microbubbles to ultrasound in a sheared flow has shown a complex coupling of ultrasound-induced volume oscillations with hydrodynamic forces: Saffman lift and the Bjerknes force. In this work, the relative influence of these two forces acting in the across-streamlines direction is determined as a function of the Reynolds and Womersley and the excitation to bubble natural frequency ratio. We use in-vitro experiments to study the behavior of microbubbles in physiologically-realistic pulsatile flows. Quantitative information about microbubble trajectories in physiological conditions is necessary to develop models in order to control ultrasound steering of bubble-based drug delivery vectors in the human vasculature.

3:20PM D11.00006 Analytical and experimental analyses of the translation of microbubbles under short acoustic pulses , ELENA IGUALADA-VILLODRE, ANA MEDINA-PALOMO, JAVIER RODRIGUEZ-RODRIGUEZ, Carlos III University of Madrid — The translation of bubbles as a result of the primary Bjerknes force is studied both analytically and experimentally. In particular, we focus on the translational dynamics of bubbles under the effect of short acoustic pulses, i.e. pulses whose duration is of the order of the characteristic viscous time based on the bubble size. The experiments developed show that existing models widely used in the literature do not allow to properly reproduce the bubble velocity history. Given the comparison between analytical and experimental results, we can conclude that the viscous drag cannot be approximated by a constant drag coefficient when the time scale is of the order of the characteristic viscous time, as the history force becomes dominant. In other words, the history force is needed to correctly reproduce the experimental results. In this talk we will show analytical solutions of the bubble translational dynamics equation for both piece-wise constant and oscillatory forcing. This work has been supported by Spanish Ministries of Science and of Economy and Competitiveness through grants: DPI2008-06369 and DPI2011-28356-C03-02.

3:33PM D11.00007 Shock-Induced Bubble Collapse in a Vessel: Implications for Vascular Injury in Shockwave Lithotripsy[1] , VEDRAN CORALIC, TIM COLONIUS, California Institute of Technology — We numerically investigate the shock-induced collapse (SIC) of a preexisting bubble in a blood vessel and evaluate the potential of such an event to contribute to onset of vascular injury in shockwave lithotripsy (SWL). Previously, we utilized a 3D, high-order accurate, shock- and interface-capturing, multicomponent flow algorithm to carry out a large-scale parametric study of this problem [V. Coralic and T. Colonius, Eur. J. Mech. B-Fluid 40, 64-74 (2013)]. The results indicated that the influence of the blood vessel on the bubble dynamics was negligible and confirmed with experiments that the vessel would freely deform under the forcing imparted by the collapse. As a result, in this study, we perform simulations of the SIC of a preexisting bubble in a free field and couple them to a freely deforming Lagrangian mesh so to characterize the deformations in the fluid surrounding the bubble, which, as our previous results suggest, may be interpreted as the vessel and surrounding tissue. We report the fully 3D and time-dependent Green-Lagrange strains and compare them to the ultimate strains obtained in uniaxial compression/tension tests in tissue. Our findings suggest that the SIC of preexisting bubbles in blood vessels is a viable mechanism by which injury may be initiated in SWL.

[1]This research was supported by NIH grant no. 2PO1DK043881 and utilized XSEDE, which is supported by NSF grant no. OCI-1053575.

3:46PM D11.00008 Dynamics of bubble collapse under vessel confinement in 2D hydrodynamic experiments[1] , GALINA SHPUNTOVA, JOANNA AUSTIN, University of Illinois at Urbana-Champaign — One trauma mechanism in biomedical treatment techniques based on the application of cumulative pressure pulses generated either externally (as in shock-wave lithotripsy) or internally (by laser-induced plasma) is the collapse of voids. However, prediction of void-collapse driven tissue damage is a challenging problem, involving complex and dynamic thermomechanical processes in a heterogeneous material. We carry out a series of model experiments to investigate the hydrodynamic processes of voids collapsing under dynamic loading in configurations designed to model cavitation with vessel confinement. The baseline case of void collapse near a single interface is also examined. Thin sheets of tissue-surrogate polymer materials with varying acoustic impedance are used to create one or two parallel material interfaces near the void. Shadowgraph photography and two-color, single-frame particle image velocimetry quantify bubble collapse dynamics including jetting, interface dynamics and penetration, and the response of the surrounding material.

[1]Research supported by NSF Award #0954769, "CAREER: Dynamics and damage of void collapse in biological materials under stress wave loading."

3:59PM D11.00009 Characterization of Acoustic Droplet Vaporization Using MRI , DAVID LI, STEVEN ALLEN, LUIS HERNANDEZ-GARCIA, JOSEPH BULL, University of Michigan — Acoustic droplet vaporization (ADV) is the selective vaporization of liquid droplets to form larger gas bubbles. The ADV process is currently being researched for biomedical applications such as gas embolotherapy, drug delivery, and phase-change contrast agents. In this study an albumin encapsulated dodecafluoropentane (DDFP, CAS: 678–26–2) microdroplet suspension was vaporized using a single element focused (f/2, D=19 mm) 3.5 MHz transducer (Panametrics A321S, Olympus, Waltham, MA). The resulting DDFP bubble clouds were imaged using both bright field microscopy and MRI (Varian 7T, Agilent Technologies Inc., Santa Clara, CA). Field distortions due to DDFP bubble generation were characterized against the bright field images as a function of acoustic power and bubble cloud size. Experimentally a direct correlation between bubble cloud dimensions generated and field distortions seen in the MRI was observed. Additionally, MR velocimetry was used to measure the flow field resulting from ADV. The field distortions due to the bubbles were further characterized by modeling Maxwell's equations using COMSOL (COMSOL Inc., Burlington, MA). The ability to characterize ADV with alternative imaging modalities may prove useful in further development of ADV based biomedical therapies.

4:12PM D11.00010 Acoustic droplet vaporization is initiated by superharmonic focusing , OLEKSANDR SHPAK, University of Twente, MARTIN VERWEIJ, Delft University of Technology, RIK VOS, NICO DE JONG, Erasmus MC, DETLEF LOHSE, MICHEL VERSLUIS, University of Twente — Acoustically sensitive emulsion microdroplets composed of a low boiling point liquid perfluorocarbon have the potential to be a highly efficient system for local drug delivery, embolotherapy or for tumor imaging. The physical mechanisms underlying the acoustic activation of these phase-change emulsions into vapor bubbles, termed acoustic droplet vaporization, have not been well understood. The droplets have a very high activation threshold, its frequency dependence does not comply with homogeneous nucleation theory and focusing spots have been observed while the wavelength is at least an order larger than the droplet size. Here we show that acoustic droplet vaporization is initiated by a combination of two effects: highly nonlinear distortion of the acoustic wave before it hits the droplet, and focusing of the distorted wave by the droplet itself. At high excitation pressures, nonlinear distortion causes significant superharmonics with wavelengths below the diameter of the droplet. The proposed model is validated with experimental data captured with an ultra high-speed camera on the exact locations of the nucleation spots. Moreover, the presented mechanism explains the hitherto counterintuitive dependence of the nucleation threshold on the ultrasound frequency.

Sunday, November 24, 2013 2:15PM - 4:25PM –

Session D12 Vortex Dynamics and Vortex Flows II 336 - Lorenz Sigurdson, University of Alberta

2:15PM D12.00001 Vortex rings impinging on porous boundaries , STUART DALZIEL, DAMTP, University of Cambridge, ANNA MUJAL COLILLES, Universitat Politecnica de Catalunya — Vortex rings and their collisions with simple, rigid boundaries have long been studied, both in their own right and as a prototype for turbulent interactions with boundaries. Over the last few years this paradigm has been extended to the impact of vortex rings on a bed of particles. Of principal interest here has been the resuspension/erosion of the particle layer. While in some parameter ranges the boundary may still appear to the vortex ring as though it is a simple rigid solid, the reality is that even if the particles do not move the boundary will be porous. Through a series of experiments, this paper explores some aspects of how the interaction between a vortex ring and a boundary is modified when the boundary is porous. The study is fundamental, and while motivated initially by the impact of a ring on sediment layers, the interaction of vortical structures and turbulence with porous boundaries has much broader applications.

2:28PM D12.00002 ABSTRACT WITHDRAWN –

2:41PM D12.00003 Simulation of the Initial 3-D Instability of an Impacting Drop Vortex Ring , LORENZ SIGURDSON, JUSTIN WIWCHAR, Vortex Fluid Dynamics Lab, Mechanical Engineering Department, University of Alberta, JENS WALTHER, Mechanical Engineering, Danish Technical University, VFDL TEAM — Computational vortex particle method simulations of a perturbed vortex ring are performed to recreate and understand the instability seen in impacting water drop experiments. Three fundamentally different initial vorticity distributions are used to attempt to trigger a Widnall instability, a Rayleigh centrifugal instability, or a vortex breakdown-type instability. Simulations which simply have a perturbed solitary ring result in an instability similar to that seen experimentally. Waviness of the core which would be expected from a Widnall instability is not visible. Adding an opposite-signed secondary vortex ring or an image vortex ring to the initial conditions, to trigger a Rayleigh or breakdown respectively, does not appear to significantly change the instability from what is seen with a solitary ring. This suggests that a Rayleigh or vortex breakdown-type instability are not likely at work, though tests are not conclusive. Perhaps the opposite-signed secondary vortex was not strong enough or placed appropriately. Elliptical streamlines , as expected, are visible in the core of the solitary ring at early times. Support from the Canadian Natural Sciences and Engineering Research Council grant 41747 is gratefully acknowledged.

2:54PM D12.00004 Amplification of Vorticity Near the Stagnation Point of Landing Gear Wheels , GRAHAM FELTHAM, ALIS EKMEKCI, University of Toronto — In this experimental investigation, a stream of steady weak vorticity impinging near the stagnation point of a landing gear wheel is shown to grow and amplify into large-scale vortices that coherently shed from the point of generation. To produce the upstream vorticity, a platinum wire of 100 micron diameter, similar to that used in hydrogen bubble visualization technique, is placed upstream of the wheel model. Experiments are conducted in a recirculating water channel. The wheel diameter is D = 152 mm. The Reynolds number based on the wire diameter is 21 and based on the wheel diameter is 32,500. Qualitative understanding of the vorticity amplification and eventual vortex shedding near the stagnation region of the wheel is achieved by employing the hydrogen bubble visualization technique while quantitative insight is collected using Particle Image Velocimetry (PIV). The size and frequency of the shed vortices are found to depend on the wheel geometry as well as the magnitude and impingement point of the inbound vorticity.

3:07PM D12.00005 Large eddy simulation of a vortex ring impinging on a bump , XI-YUN LU, HENG REN, University of Science and Technology of China — Large eddy simulation of a vortex ring impacting a three-dimensional bump has been carried out for different bump heights and vortex core thicknesses related to thin and thick vortex rings. Various fundamental mechanisms dictating the flow behaviors, including the dynamics and instability of vortex ring, the evolution of vortical structures, and the flow transition from laminar to turbulent state, have been studied systematically. Based on the analysis of the evolution of vortical structures, the formations of loop-like vortices wrapping around the primary and secondary vortex rings and the hair-pin vortices due to the severe distortion of the secondary ring are investigated. The circulation of the vortex ring reasonably elucidates some typical phases of flow evolution. Further, the analysis of turbulent kinetic energy reveals the transition from laminar to turbulent state.

3:20PM D12.00006 Interaction of a Vortex Ring Parallel to a Plane Wall[1] , MITCHELL ALBRECHT, DOUGLAS BOHL, Clarkson University — In this work, Laser Induced Fluorescence (LIF) is used to investigate the motion of a vortex ring parallel to a plane wall. When the wall is more than 1.75 generator diameters (D_{gen}) away from the center of the generator, there is no observed effect on the path of the vortex ring. When the wall is closer, the vortex ring initially convects parallel to the wall and then turns towards the wall. The location at which the ring begins to turn towards the wall is a function of the wall location. This motion is consistent with inviscid theory. For moderate distance (1.75 D_{gen} to 0.75 D_{gen}) both legs of the vortex ring break up before interacting with the wall. When the wall is very close to the vortex ring (<0.75 D_{gen}), the leg of the vortex ring closest to the wall first moves towards, then bounces and moves away from the wall. Meanwhile, the leg farthest from the wall continues towards the wall and interacts, forming boundary layer and new shed structures. This process is qualitatively similar to the interaction of a vortex ring normal to a plane wall.

[1]This work supported by NSF Grant # 0845882.

3:33PM D12.00007 Numerical study of vorticity-enhanced heat transfer , XIAOLIN WANG, Georgia Tech, SILAS ALBEN, University of Michigan — Vortices produced by vibrated reeds and flapping foils can improve heat transfer efficiency in electronic hardware. Vortices enhance forced convection by boundary layer separation and thermal mixing in the bulk flow. In this work, we modeled and simulated the fluid flow and temperature in a 2-D channel flow with vortices injected at the upstream boundary. We classified four types of vortex streets depending on the Reynolds number and vortices' strengths and spacings, and studied the different vortex dynamics in each situation. We then used Lagrangian coherent structures (LCS) to study the effect of the vortices on mixing and determined how the Nusselt number and Coefficients of performance vary with flow parameters and Peclet numbers.

3:46PM D12.00008 Controlling vortex breakdown in swirling pipe flows: experiments and simulations

, DAVID J.C. DENNIS, University of Liverpool, CHRISTOPHE SERAUDIE, University of Liverpool and Ecole Centrale de Nantes, ROBERT J. POOLE, University of Liverpool — A laminar, incompressible, viscous pipe flow with a controllable wall swirl has been studied both numerically and experimentally across a Reynolds number range of 2 to 30. The pipe consists of two smoothly joined sections that can be rotated independently about the same axis. The circumstances of flow entering a stationary pipe from a rotating pipe (decaying swirl) and flow entering a rotating pipe from a stationary pipe (growing swirl) have been investigated. Flow visualisations show that at a certain swirl ratio, which can be different for growing and decaying swirl at the same Reynolds number, vortex breakdown occurs. The variation of this critical swirl ratio with Reynolds number is explored and good agreement is found between the experimental and numerical methods. At high Re the critical swirl ratio tends to a constant value, whereas at low Re the product of the Reynolds number and the square of the swirl ratio tends to a constant value in agreement with an existing analytical solution. For decaying swirl the vortex breakdown manifests itself on the pipe axis, whereas for growing swirl it forms near the pipe wall. The vortex flow formed at critical conditions is found to increase radially and axially with increasing Reynolds number and swirl ratio.

3:59PM D12.00009 Flow instability and vortex street in eccentric annular channels[1]

, GEORGE CHOUEIRI, STAVROS TAVOULARIS, University of Ottawa — Flow development in an eccentric annular channel with a diameter ratio of 0.5 has been investigated using flow visualization, two-component laser Doppler velocimetry and planar and stereoscopic particle image velocimetry. The eccentricity e was varied between 0.3 and 0.9 and the Reynolds number was $1000 \leq \mathrm{Re} \leq 18000$. For sufficiently large e and Re, large differences developed between the velocity in the gap region and the one in the rest of the channel; these were accompanied by flow instability and the generation of a quasi-periodic vortex street, which manifested itself by strong cross-flows across the gap and an increase in axial velocity in the gap region, but also affected the flow in the entire channel. The vortex strength was highest for $e \approx 0.7$ and the Strouhal number of the cross-flow oscillations (based on bulk velocity and core diameter) increased with increasing Re, reaching an asymptote near 0.12 for $\mathrm{Re} \geq 10000$.

[1]Supported by NSERC and AECL

4:12PM D12.00010 Stability Analysis of the Vortex Rope Formed in Draft Tubes

, GIRISH KUMAR RAJAN, JOHN CIMBALA, The Pennsylvania State University — Studies on draft tube surge have shown that there are undesirable effects in the form of violent pressure fluctuations caused by a helical vortex (often called the vortex rope), formed in the draft tube due to a shear layer produced by a central stalled region with lesser axial velocities, and the swirling main-flow. The vortex rope is formed when hydroturbines operate away from the best efficiency point, and affects the efficiency of the turbine severely. Thus, in order to reduce these undesired effects of the vortex rope, a proper understanding of its structure and stability is necessary. This project, which is in progress, involves a numerical investigation of the vortex rope and its elimination, and a mathematical analysis that could possibly throw some light on the stability of the rope. Several cases have been simulated in ANSYS-FLUENT with the draft tube geometry obtained from the FLINDT project. It is then possible to obtain the vortex rope parameters as functions of the discharge coefficient. In addition, the simulations are also expected to provide information on the mean velocity field in the draft tube. These relations might also be of some help in the stability analysis, which should identify the modes that are unstable.

Sunday, November 24, 2013 2:15PM - 4:12PM –

Session D13 Focus Session: Marine Hydrokinetic Energy Conversion II 301 - Martin Wosnik, University of New Hampshire

2:15PM D13.00001 Real-time Ocean Wave Prediction for Optimal Performance of a Wave Energy Converter

, DANIELE CAVAGLIERI, THOMAS BEWLEY, University of California San Diego — In recent years, there has been a growing interest in renewable energy. Among all the available possibilities, wave energy conversion, due to the huge availability of energy that the ocean could provide, represents nowadays one of the most promising solutions. However, the efficiency of a wave energy converter for ocean wave energy harvesting is still far from making it competitive with more mature fields of renewable energy, such as solar and wind energy. One of the main problems is related to the inability to accurately predict the profile of oncoming waves approaching the wave energy converter. For this reason, we developed a new hybrid method for state estimation of nonlinear systems, which is based on a variational formulation of an ensemble smoother, combined with the formulation of the ensemble Kalman smoother. This method has been employed for the optimal forecasting of ocean waves via sensors placed on an array of wave energy converters. The coupled simulation of ocean waves and energy devices has been carried out leveraging a nonlinear High Order Spectral code.

2:28PM D13.00002 Model Scaling of Hydrokinetic Ocean Renewable Energy Systems[1]

, KARL VON ELLENRIEDER, WILLIAM VALENTINE, Florida Atlantic University — Numerical simulations are performed to validate a non-dimensional dynamic scaling procedure that can be applied to subsurface and deeply moored systems, such as hydrokinetic ocean renewable energy devices. The prototype systems are moored in water 400 m deep and include: subsurface spherical buoys moored in a shear current and excited by waves; an ocean current turbine excited by waves; and a deeply submerged spherical buoy in a shear current excited by strong current fluctuations. The corresponding model systems, which are scaled based on relative water depths of 10 m and 40 m, are also studied. For each case examined, the response of the model system closely matches the scaled response of the corresponding full-sized prototype system. The results suggest that laboratory-scale testing of complete ocean current renewable energy systems moored in a current is possible.

[1]This work was supported by the U.S. Southeast National Marine Renewable Energy Center (SNMREC)

2:41PM D13.00003 ABSTRACT WITHDRAWN –

2:54PM D13.00004 Effect of flow rate and concentration difference on reverse electrodialysis system[1]

, KILSUGN KWON, JAESUK HAN, DAEJOONG KIM, Department of Mechanical Engineering, Sogang University — Various energy conversion technologies have been developed to reduce dependency on limited fossil fuels, including wind power, solar power, hydropower, ocean power, and geothermal power. Among them, reverse electrodialysis (RED), which is one type of salinity gradient power (SGP), has received much attention due to high reliability and simplicity without moving parts. Here, we experimentally evaluated the RED performance with several parameters like flow rate of concentrated and dilute solution, concentration difference, and temperature. RED was composed of endplates, electrodes, spacers, anion exchange membrane, and cation exchange membrane. Endplates are made by a polypropylene. It included the electrodes, flow field for the electrode rinse solution, and path to supply a concentrated and dilute solution. Titanium coated by iridium and ruthenium was used as the electrode. The electrode rinse solution based on hexacyanoferrate system is used to reduce the power loss generated by conversion process form ionic current to electric current. Maximum power monotonously increases as increasing flow rate and concentration difference. Net power has optimal point because pumping power consumption increases with flow rate.

[1]This work was supported by Basic Science Research Program (Grat No. NRF-2011-0009993) through the National Research Foundation of Korea.

3:07PM D13.00005 Investigation of Energy Harvesting Using Flapping Foils , AMIN MIVEHCHI, AMANDA PERSICHETTI, BRANDON DUNHAM, JASON M. DAHL, Department of Ocean Engineering , University of Rhode Island — When harvesting kinetic energy using a flapping foil, the separation of coherent structures in the wake is crucial for determining forces on the body. Applications for utilizing energy harvesting with a flapping foil include powering of local, low power equipment and recharging AUV batteries that use flapping foils for propulsion and maneuvering. In each of these cases, it is critical to accurately predict the physical behavior and location of vortices in relation to the motion of the body in order to maximize energy output. A two-dimensional open source boundary data immersion method (LilyPad) is used for simulating the flapping motion of a foil for energy harvesting in a current. Forced motion of the flapping body indicates theoretical efficiencies for energy harvesting near 43 percent under specific flapping conditions. A simple control scheme based on pressure sensing on the surface of the foil is developed to control pitch of the foil while energy harvesting occurs in the heave direction. The control scheme is tested through real time numerical simulation. Comparisons are made with physical laboratory experiments, demonstrating high efficiencies in energy harvesting.

3:20PM D13.00006 The Impact of Blade Roughness and Biofouling on the Performance of a Horizontal Axis Marine Current Turbine[1] , KAREN FLACK, United States Naval Academy, JESSICA WALKER, Australian Maritime College, MICHAEL SCHULTZ, ETHAN LUST, United States Naval Academy — The impact of blade roughness and biofouling on the performance of a two-bladed horizontal axis marine current turbine was investigated experimentally and numerically. A 0.8 m diameter rotor (1/25th scale) with a NACA 63-618 cross section was tested in a towing tank. The torque, thrust and rotational speed were measured in the range 5 $<\lambda<$11 (λ = tip speed ratio). Three different cases were tested: clean blades, artificially fouled blades and roughened blades. The performance of the turbine was predicted using Blade Element Momentum theory and validated using the experimental results. The lift and drag curves necessary for the numerical model were obtained by testing a 2D NACA 63-618 airfoil in a wind tunnel under clean and roughened conditions. The numerical model predicts the trends that were observed in the experimental data for roughened blades. The artificially fouled blades did not adversely affect turbine performance, as the vast majority of the fouling sheared off. For the case of roughened blades, the power coefficient (C_P) versus λ curve was significantly offset below that for the clean case. The maximum C_P for this condition was 0.34, compared to 0.42 for the clean condition.

[1]Work supported by Australian Fulbright Association and ONR.

3:33PM D13.00007 The Influence of depth and surface waves on marine current turbine performance[1] , ETHAN LUST, KAREN FLACK, LUKSA LUZNIK, MAX VAN BENTHEM, U.S. Naval Academy, JESSICA WALKER, University of Tasmania — Performance characteristics are presented for a $1/25^{\text{th}}$ scale marine current turbine operating in calm conditions and in the presence of intermediate and deep water waves. The two-bladed turbine has radius of 0.4 m and a maximum blade pitch of 17°. The hydrofoil is a NACA63-618 which was selected to be Reynolds number independent for lift in the operational range (Re_C = 2 - 4 x 10^5). The experiments were performed in the 116 m tow-tank at the United States Naval Academy at depths of 0.8D and 1.75D measured from the blade tip to the mean free surface. Overall average values for power and thrust coefficient were found to be insensitive to wave form and weakly sensitive to turbine depth. Waves yield a small increase in turbine performance which can be explained by Stokes drift. Variations on performance parameters are on the same order of magnitude as the average value especially near the mean free surface and in the presence of high energy waves.

[1]Office of Naval Research

3:46PM D13.00008 Characterizing Turbulent Events at a Tidal Energy Site from Acoustic Doppler Velocity Observations , KATHERINE MCCAFFREY[1], University of Colorado at Boulder, BAYLOR FOX-KEMPER[2], Brown University, PETER HAMLINGTON[3], University of Colorado at Boulder — As interest in marine renewable energy increases, observations are crucial to understanding the environments encountered by energy conversion devices. Data obtained from an acoustic Doppler current profiler and an acoustic Doppler velocimeter at two locations in the Puget Sound, WA are used to perform a detailed analysis of the turbulent environment that is expected to be present at a turbine placed in a tidal strait. Metrics such as turbulence intensity, structure functions, probability density functions, intermittency, coherent turbulence kinetic energy, anisotropy invariants, and linear combinations of eigenvalues are used to characterize the turbulence. The results indicate that coherent turbulence kinetic energy and turbulence intensity can be used to identify and parameterize different turbulent events in the flow. An analysis of the anisotropy characteristics leads to a physical description of turbulent events (defined using both turbulence intensity and coherent turbulent kinetic energy) as being dominated by one component of the Reynolds stresses. During non-turbulent events, the flow is dominated by two Reynolds stress components. The importance of these results for the development of realistic models of energy conversion devices is outlined.

[1]Cooperative Institute for Research in Environmental Sciences, Department of Atmospheric and Oceanic Sciences
[2]Department of Geological Sciences
[3]Department of Mechanical Engineering

3:59PM D13.00009 Experimental Investigation of Effects of Blockage and Free Surface Proximity on Flow-field and Performance of a Hydrokinetic Turbine[1] , NITIN KOLEKAR, ARINDAM BANERJEE, Lehigh University — Results from an experimental study to investigate the effect of blockage and free surface proximity on the performance of a constant chord, zero twist, fixed pitch hydro kinetic turbine in an open surface water channel will be presented. The presence of free surface and the size of turbine relative to the flow channel (blockage effect) affects the fluid dynamics around and in the near wake of turbine and hence the thrust-torque loading on turbine blades. Detailed parametric studies will be carried out to understand the effect of free surface proximity, Froude number (which depends on water velocity and depth of the channel), turbine proximity to channel walls and blockage on the turbine performance. Characterization of wake meandering and flow around the turbine is performed using a stereo-Particle Image Velocimetry technique for flows with various Froude number. The thrust and torque on turbine will be measured using a submerged thrust-torque sensor in-line with the turbine. The results of experiments will be compared with analytical models based on blade element momentum theory by modeling free surface and blockage effects.

[1]The authors gratefully acknowledge the financial support from the Office of Naval Research through contract ONR N000141010923.

Sunday, November 24, 2013 2:15PM - 4:12PM –

Session D14 Experimental Techniques II: Aerodynamics/Wind Tunnel 302 - Thomas Corke, University of Notre Dame

2:15PM D14.00001 Flow measurements on a low speed wind tunnel , DIANA GARCIA, ROBERTO MARTINEZ, Universidad Nacional de Colombia — The design and performance of a wind tunnel are tested with its characterization, that implies a detailed study of the pressure losses and the velocity profiles, which will provide knowledge of the behaviour of the flow in the tunnel. Among the results achieved, it was obtained a maximum velocity of 72 Km/h in the test section, also it was determined that its essential to implement various improvements in the design of the tunnel. It is found through the present that it is not possible to calculate the net loss in the tunnel. However, having into an account experimental parameters of the construction of the tunnel one can estimate a percentage of loss flow comparing it to the one calculated in the test section. It must also be mentioned that the losses found relative to the test section and the diffuser are negligible compared to the one relative to the contractor. This suggest a number of improvements in the design of the tunnel, mainly in the joints between sections.

2:28PM D14.00002 Fast response temperature and humidity sensors for measurements in high Reynolds number flows[1] , YUYANG FAN, GILAD ARWATZ, MARGIT VALLIKIVI, MARCUS HULTMARK, Princeton University — Conventional hot/cold wires have been widely used in measuring velocity and temperature in turbulent flows due to their fine resolutions and fast response. However, for very high Reynolds number flows, limitations on the resolution appear. A very high Reynolds number flow is the atmospheric boundary layer. In order to accurately predict the energy balance at the Earth's surface, one needs information about the different turbulent scalar fields, mainly temperature and humidity, which together with velocity, contribute to the turbulent fluxes away from the surface. The nano-scaled thermal anemometry probe (NSTAP) was previously developed at Princeton and has proven to have much higher spatial and temporal resolution than the regular hot wires. Here we introduce new fast-response temperature and humidity sensors that have been developed and tested. These sensors are made in-house using standard MEMS manufacturing techniques, leaving high flexibility in the process for optimization to different conditions. The small dimensions of these novel sensors enable very high spatial resolution while the small thermal mass allows significant improvements in the frequency response. These sensors have shown promising results in acquiring un-biased data of turbulent scalar and vector fields.

[1]Supported under ONR Grants N00014-12-1-0875 and N00014-12-1-0962 (program manager Ki-Han Kim)

2:41PM D14.00003 Dynamic calibration and modeling of a cold wire for temperature measurement[1] , GILAD ARWATZ, CARLA BAHRI, Princeton University, ALEXANDER SMITS, Princeton University, Princeton NJ 08544, U.S.A and Monash University, VIC 3800, Australia, MARCUS HULTMARK, Princeton University — The dynamical behavior of cold wires and their supporting structure is investigated. It is shown that conventional cold wires are slower than previously believed, which can cause substantial inaccuracies in temperature data. A new method for accurate temperature measurements using cold wires is presented. A lumped parameter model that accounts for the effects of end conduction on wire response is developed. To verify and validate the model, an experimental investigation is conducted where the frequency response of the wire is investigated under different heating conditions. The response obtained from the model is compared to experimental results with convincing agreement. The new model can be used to correct data acquired with conventional cold wire sensors, with non-negligible end-conduction effects, so that accurate measurements can be obtained. Lastly, the new model can be used to design a new temperature sensor that has a better frequency response when compared to conventional cold-wires.

[1]Funding provided by the office of Naval Research (Grant #: N00014-12-1-0875).

2:54PM D14.00004 Hot-wire based phase resolved measurement techniques for turbomachinery flows , NICHOLAS JAFFA, SCOTT MORRIS, JOSHUA CAMERON, University of Notre Dame — Resolving the details of turbomachinery rotor flows from the stationary reference frame is difficult due to the high sensor frequency response required. Hot-wires have the necessary frequency response but are sensitive to both total temperature and velocity. In high-speed turbomachinery flows, the large blade-to-blade total temperature gradients prevent traditional hot-wire methods from being used to measure velocity directly. In order to decouple the effects from the temperature variation, a single constant temperature hot-wire was operated at different overheats at the exit of a high-speed transonic axial compressor rotor. The multiple overheat method was used to decouple the phase locked averages of total temperature and velocity magnitude from the phase locked average hot-wire voltages for different overheats. The phase locked average total temperature and velocity magnitude fields show flow features relative to the rotor including blade wakes, boundary layers, and tip clearance flows.

3:07PM D14.00005 Development of a Digital Image Projection (DIP) Technique to Quantify Wind Driven Water Droplet/Rivulet Flows over a NACA 0012 Airfoil[1] , HUI HU, KAI ZHANG, Iowa State University — A digital image projection (DIP) technique is developed to achieve non-intrusive thickness measurements of wind-driven water droplet/rivulet flows. The DIP technique is based on the principle of structured light triangulation in a similar manner as a stereo vision system but replacing one of the cameras for stereo imaging with a digital projector. A grid pattern of known characteristics is projected onto a test object (i.e., the droplet/rivulet over the test plate). Due to 3D shape profile of the test object, the projected grid pattern is deformed seen from a perspective different from the projection axis. By comparing the distorted image over the test object and a reference image, the 3D profile of the test object with respect to the reference plane (i.e., the thickness distribution of the droplet/rivulet flow) can be retrieved quantitatively and instantaneously. The DIP system is used to quantify the dynamic shape change and stumbling runback motion of the wind-driven water droplet/rivulet flows over a NACA0012 airfoil. Such information is highly desirable to elucidate the underlying physics to improve our understanding about the surface water transport process pertinent to ice formation and accretion over aircraft wings in atmospheric icing conditions.

[1]The research work is funded by NSF and NASA.

3:20PM D14.00006 Plasma Anemometer Measurements and Optimization[1] , CURTIS MARSHALL, ERIC MATLIS, THOMAS CORKE, University of Notre Dame, SIVARAM GOGINENI, Spectral Energies, LLC — Velocity measurements using a constant-current plasma anemometer were performed in a Mach 0.4 jet in order to further optimize the anemometer design. The plasma anemometer uses an AC glow discharge (plasma) formed in the air gap between two protruding low profile electrodes as the flow sensing element. The output from the anemometer is an amplitude modulated version of the AC voltage input that contains information about the mean fluctuating velocity components. Experiments were performed to investigate the effect of the electrode gap, AC current, and AC frequency on the mean and fluctuating velocity sensitivity and repeatability of the sensor. This involved mean velocity calibrations from 0 to 140 m/s and mean and fluctuating velocity profiles through the shear layer of the jet. Measurements with a constant temperature hot-wire anemometer were used for reference. The results showed an improvement in performance with increasing AC frequency that was attributed a more stable glow discharge. The agreement with the hot-wire were good, with the advantage of the plasma anemometer being its 100-times higher frequency response.

[1]Supported by Air Force SBIR Phase II FA8650-11-C-2199

3:33PM D14.00007 Simultaneous time-resolved measurement of flow field and surface deformation combining tomographic PIV and Mach-Zehnder interferometry[1] , CAO ZHANG, RINALDO MIORINI, JOSEPH KATZ, Johns Hopkins University — Flow induced vibrations are ubiquitous in numerous applications, for which knowledge of the relationship between surface deformation and the corresponding flow field is vital to the understanding of the processes involved. A novel technique, combining tomographic PIV (TPIV) and Mach-Zehnder interferometry (MZI), has been developed to perform simultaneous, time-resolved measurements of both the 3D flow field above a complaint transparent PDMS wall, and the spatial distribution of surface deformation. Five high speed cameras are involved, four for TPIV, and the fifth for MZI. The same high-speed laser is used for both measurements by allowing a small fraction (0.1%) of the TPIV light, which is transmitted through PDMS coating, to propagate through a 99.9% mirror to the MZI camera. This object beam interferes with a similarly weak reference beam that does not pass through the sample volume. Methods for extracting the deformation from the resulting fringes will be discussed, such as fringe enhancement techniques to improve the S/N ratio. Sample velocity and deformation results recorded at 3kHz will be presented, demonstrating the ability of combined TPIV and MZI to study the dynamical interactions between 3D flow structure and surface deformation.

[1]Sponsored by ONR and NSF-MRI.

3:46PM D14.00008 Single Rod Vibration in Axial Flow , NOAH WEICHSELBAUM, SHENGFU WANG, PHILIPPE BARDET, The George Washington University — Fluid structure interaction of a single rod in axial flow is a coupled dynamical system present in many application including nuclear reactors, steam generators, and towed antenna arrays. Fluid-structure response can be quantified thanks to detailed experimental data where both structure and fluid responses are recorded. Such datum deepen understanding of the physics inherent to the system and provide high-dimensionality quantitative measurements to validate coupled structural and CFD codes with various level of complexity. In this work, single rods fixed on both ends in a concentric pipe, are subjected to an axial flow with Reynolds number based on hydraulic diameter of Re =4000 . Rods of varying material stiffness and diameter are utilized in the experiment resulting in a range of dimensionless U between 0.5 and 1, where $U = (\rho A/EI)^{1/2} uL$. Experimental measurements of the velocity field around the rod are taken with PIV from time-resolved Nd:YLF laser and a high speed CMOS camera. Three-dimensional and temporal vibration and deflection of the rod is recorded with shadowgraphy utilizing two sets of pulsed high power LED and dedicated CMOS camera. Through integration of these two diagnostics, it is possible to reconstruct the full FSI domain providing unique validation data.

3:59PM D14.00009 Lonely GPFUTV–the movement of water under the action of unknown vacuum[1] , WEIYI LIN[2], Independent Investigator of GPFUTV — In this paper, firstly, the experiment on the flow resistance of the aerated pipe flow is introduced. The experimental research on comparison between different volumes of air entrained is presented. Secondly, the characteristics of gravity pipe flow under the action of Torricelli's vacuum, shortly called as GPFUTV are dissertated, including creative and functional design, fundamental principle, etc. Under the joint action of an unknown vacuum energy and the formation of non-aerated flow the water flow is full-pipe and continuous, high-speed and non-rotational as distinguished from turbulent flow. Thirdly, an appeal in relation to the experimental research, the applied studies and basic theory research is given. For instance, experimental study of Torricelli's experiment phenomenon in the vacuum environment, applied study of the potential for GPFUTV to be developed for deep seawater suction technology and lifting technology for deep ocean mining, theoretical study of flow stability and flow resistance under GPFUTV condition, etc. At last, the famous GPFUTV project is illustrated.

[1]12 years of rigorous and independent survey research

[2]I have spent most of the working hours in a survey of research on GPFUTV in the past twelve years. GPFUTV technology itself contains relevant applied research and basic research, which for classical Fluid Mechanics will be a revolutionary contribution.

Sunday, November 24, 2013 2:15PM - 4:25PM –

Session D16 Biofluids: Physiological II - Computational Blood Flow in Arteries 304 - Paolo Zunino, University of Pittsburgh

2:15PM D16.00001 Modeling blood flow as a fluid- multilayered structure interaction problem consisting of poroelastic materials , MARTINA BUKAC, PAOLO ZUNINO, IVAN YOTOV, University of Pittsburgh — We model arterial blood flow as an incompressible Newtonian fluid confined by a multilayered poroelastic wall. We consider a two layer model for the arterial wall, where the inner layers (the endothelium and the intima) behave as a thin structure modeled as a linearly elastic Koiter membrane, while the outer part of the artery (the media and adventitia) is described by the Biot model. The fluid, membrane, and poroelastic structure are two-way coupled via kinematic and dynamic coupling conditions. We propose and analyze a splitting strategy based on the Lie operator splitting method, which allows solving the Navier-Stokes and Biot equations separately. In this way, we uncouple the original problem into two problems defined on separate subregions, the lumen and the wall. We show that the proposed scheme is stable under mild restriction of the time approximation step. Numerically, we investigate the effects of porosity to the structure displacement. We distinguish a high storativity and a high permeability case in the Darcy equations, and compare them to the results obtained using a purely elastic model. Depending on the regime, we observe significantly different behaviors in response to perturbations of each parameter.

2:28PM D16.00002 Toward non-Newtonian effects on secondary flow structures in a 180 degree bent tube model for curved arteries , STEVIN VAN WYK, LISA PRAHL WITTBERG, LASZLO FUCHS, Linne FLOW Center, KTH, Stockholm, Sweden, KARTIK V. BULUSU, MICHAEL W. PLESNIAK, The George Washington University — The purpose of this study is to investigate the development of vortical flow structures of blood like fluids in a 180 degree tube bend, analogous to the aortic arch. Cardiovascular diseases are localized to regions of curvature in the arterial tree. The pathology of atherogenesis is widely considered an inflammatory response, hypothesized to be modulated by the interplay between Wall Shear Stress (WSS) variations and particulate transport mechanisms from the bulk fluid core to the near wall. The WSS is determined by the local flow characteristics as well as the rheological properties of the blood, which in turn are dependent on the bulk secondary flows. In this work, the time dependent fluid flow under various physiological flow conditions are investigated both experimentally and numerically. A Newtonian blood analog fluid model is considered in both studies to validate both methods and thereby study flow structure development during steady as well as pulsatile conditions. Particle image velocimetry (2C – 2D PIV) is used to acquire velocity field data from an acrylic tube bend. The numerical study is extended to consider the non-Newtonian properties of blood according to an empirical model to identify the relative importance of the non-Newtonian behavior. The studies show complex Dean and Lyne vortex interaction that are enhanced with increasing peak Reynolds numbers.

2:41PM D16.00003 Hydrodynamic Enhancements of Dissolution from Drug Particles: *In vivo* vs. *In vitro* , JAMES BRASSEUR, Penn State, YANXING WANG, Georgia Tech — Absorption of drug molecules into the blood stream is generally limited by dissolution-rate in the intestines. Dissolution occurs via diffusion enhanced by a response to the hydrodynamic flow environment, a process that is very different in the human intestine than in a USP-II dissolution apparatus, commonly used by drug companies to validate new drug formulations. Whereas *in vivo* hydrodynamics are driven by the motility of intestinal wall muscles, the USP-II apparatus is a rotating paddle to mix drug particles during dissolution testing. These differences are of current interest to agencies that regulate drug product development. Through lattice-Boltzmann-based computer simulation of point particles transported through human intestine, we analyze the hydrodynamic parameters associated with convection that quantify the extent to which *in vitro* dissolution tests are or are not relevant to *in vivo* hydrodynamics. . We show that for drug particles less that ∼100-200 microns, effects of convection are negligible in the intestines. However, we discover a previously unappreciated phenomenon that significantly enhances dissolution-rate and that distinguishes *in vitro* from *in vivo* dissolution: the fluid shear rate at the particle. *Supported by NSF and AstraZeneca.*

2:54PM D16.00004 Analysis of perfusion, microcirculation and drug transport in tumors. A computational study. , PAOLO ZUNINO, Department of Mechanical Engineering and Materials Science, University of Pittsburgh, LAURA CATTANEO, MOX, Department of Mathematics, Politecnico di Milano — We address blood flow through a network of capillaries surrounded by a porous interstitium. We develop a computational model based on the Immersed Boundary method [C. S. Peskin. Acta Numer. 2002.]. The advantage of such an approach relies in its efficiency, because it does not need a full description of the real geometry allowing for a large economy of memory and CPU time and it facilitates handling fully realistic vascular networks [L. Cattaneo and P. Zunino. Technical report, MOX, Department of Mathematics, Politecnico di Milano, 2013.]. The analysis of perfusion and drug release in vascularized tumors is a relevant application of such techniques. Blood vessels in tumors are substantially leakier than in healthy tissue and they are tortuous. These vascular abnormalities lead to an impaired blood supply and abnormal tumor microenvironment characterized by hypoxia and elevated interstitial fluid pressure that reduces the distribution of drugs through advection [L.T. Baxter and R.K. Jain. Microvascular Research, 1989]. Finally, we discuss the application of the model to deliver nanoparticles. In particular, transport of nanoparticles in the vessels network, their adhesion to the vessel wall and the drug release in the surrounding tissue will be addressed.

3:07PM D16.00005 Fractional-order viscoelasticity in one-dimensional blood flow models[1] , PARIS PERDIKARIS, GEORGE KARNIADAKIS, Brown University, CRUNCH GROUP TEAM — In this work, we have integrated different integer, and for the first time, fractional order viscoelastic models in a one-dimensional blood flow solver, and we study their behavior by presenting an in-silico study on a patient-specific arterial network. Integer-order models are directly derived from the QLV (quasi linear viscoelasticity) theory and are comprised by simple combinations of springs and dashpots. Fractional-order models employ fractional derivatives and naturally introduce a new element, the so called "spring-pot." We perform one-dimensional blood flow simulations in a large patient-specific cranial network using four different viscoelastic parameter data-sets. The results aim to quantify the effect of arterial wall viscoelasticity on pulse wave propagation, as well as reflect any sensitivity on the input parameters that define each model. To this end, we provide a comparison of several viscoelastic models, highlight the important role played by the fractional order, and carry out a detailed global sensitivity analysis study on a stochastic fractional order viscoelastic model.

[1]This work was supported by the DOE Collaboratory on Mathematics for Mesoscopic Modeling of Materials (CM4) and the DOE/INCITE program.

3:20PM D16.00006 A Porous Media Model for Blood Flow within Reticulated Foam , JASON ORTEGA, Lawrence Livermore National Laboratory — A porous media model is developed for non-Newtonian blood flow through reticulated foam at Reynolds numbers ranging from 10^{-8} to 10. This empirical model effectively divides the pressure gradient versus flow speed curve into three regimes, in which either the non-Newtonian viscous forces, the Newtonian viscous forces, or the inertial fluid forces are most prevalent. When compared to simulation data of blood flow through two reticulated foam geometries, the model adequately captures the pressure gradient within all three regimes, especially that within the Newtonian regime where blood transitions from a power-law to a constant viscosity fluid. This work was supported by the National Institutes of Health/National Institute of Biomedical Imaging and Bioengineering Grant R01EB000462 and partially performed under the auspices of the U.S. Department of Energy by Lawrence Livermore National Laboratory under Contract DE-AC52-07NA27344.

3:33PM D16.00007 Vortex dynamics in ruptured and unruptured intracranial aneurysms , GABRIEL TRYLESINSKI, University at Buffalo, State University of New York; ISAE-ENSICA, Toulouse, France, NICOLE VARBLE, JIANPING XIANG, HUI MENG, University at Buffalo, The State University of New York — Intracranial aneurysms (IAs) are potentially devastating pathological dilations of arterial walls that affect 2-5% of the population. In our previous CFD study of 119 IAs, we found that ruptured aneurysms were correlated with complex flow pattern and statistically predictable by low wall shear stress and high oscillatory shear index. To understand flow mechanisms that drive the pathophysiology of aneurysm wall leading to either stabilization or growth and rupture, we aim at exploring vortex dynamics of aneurysmal flow and provide insight into the correlation between the previous predictive morphological parameters and wall hemodynamic metrics. We adopt the Q-criterion definition of coherent structures (CS) and analyze the CS dynamics in aneurysmal flows for both ruptured and unruptured IA cases. For the first time, we draw relevant biological conclusions concerning aneurysm flow mechanisms and pathophysiological outcome. In pulsatile simulations, the coherent structures are analyzed in these 119 patient-specific geometries obtained using 3D angiograms. The images were reconstructed and CFD were performed. Upon conclusion of this work, better understanding of flow patterns of unstable aneurysms may lead to improved clinical outcome.

3:46PM D16.00008 Accuracy of Computational Cerebral Aneurysm Hemodynamics Using Patient-Specific Endovascular Measurements[1] , PATRICK MCGAH, MICHAEL LEVITT, MICHAEL BARBOUR, PIERRE MOURAD, LOUIS KIM, ALBERTO ALISEDA, University of Washington — We study the hemodynamic conditions in patients with cerebral aneurysms through endovascular measurements and computational fluid dynamics. Ten unruptured cerebral aneurysms were clinically assessed by three dimensional rotational angiography and an endovascular guidewire with dual Doppler ultrasound transducer and piezoresistive pressure sensor at multiple peri-aneurysmal locations. These measurements are used to define boundary conditions for flow simulations at and near the aneurysms. The additional in vivo measurements, which were not prescribed in the simulation, are used to assess the accuracy of the simulated flow velocity and pressure. We also performed simulations with stereotypical literature-derived boundary conditions. Simulated velocities using patient-specific boundary conditions showed good agreement with the guidewire measurements, with no systematic bias and a random scatter of about 25%. Simulated velocities using the literature-derived values showed a systematic over-prediction in velocity by 30% with a random scatter of about 40%. Computational hemodynamics using endovascularly-derived patient-specific boundary conditions have the potential to improve treatment predictions as they provide more accurate and precise results of the aneurysmal hemodynamics.

[1]Supported by an R03 grant from NIH/NINDS

3:59PM D16.00009 Effects of Aortic Irregularities on the Blood Flow , IRIS GUTMARK-LITTLE, Cincinnati Childrens Medical Center, LISA PRAHL-WITTBERG, STEVIN VAN WYK, MIHAI MIHAESCU, LASZLO FUCHS, Royal Institute of Technology, Sweden, PHILIPPE BACKELJAUW, Cincinnati Childrens Medical Center, EPHRAIM GUTMARK, University of Cincinnati — Cardiovascular defects characterized by geometrical anomalies of the aorta and its effect on the blood flow are investigated. The flow characteristics change with the aorta geometry and the rheological properties of the blood. Flow characteristics such as wall shear stress often play an important role in the development of vascular disease. In the present study, blood is considered to be non-Newtonian and is modeled using the Quemada model, an empirical model that is valid for different red blood cell loading. Three patient-specific aortic geometries are studied using Large Eddy Simulations (LES). The three geometries represent malformations that are typical in patients populations having a genetic disorder called Turner syndrome. The results show a highly complex flow with regions of recirculation that are enhanced in two of the three aortas. Moreover, blood flow is diverted, due to the malformations, from the descending aorta to the three side branches of the arch. The geometry having an elongated transverse aorta has larger areas of strong oscillatory wall shear stress.

4:12PM D16.00010 A Computational Fluid Dynamic Study of Blood Flow Within the Coiled Umbilical Arteries , DAVID WILKE, The University of Adelaide, JAMES DENIER, The University of Auckland, TRENT MATTNER, YEE KHONG, The University of Adelaide — The umbilical cord is the lifeline of the fetus throughout gestation. In a normal pregnancy it facilitates the supply of oxygen and nutrients from the placenta via a single vein, in addition to the return of deoxygenated blood from the developing embryo or fetus via two umbilical arteries. Despite the major role it plays in the growth of the fetus, pathologies of the umbilical cord are poorly understood. In particular, variations in the cord geometry, which typically forms a helical arrangement, have been correlated with adverse outcomes in pregnancy. Cords exhibiting either abnormally low or high levels of coiling have been associated with pathological results including growth-restriction and fetal demise. Despite this, the methodology currently employed by clinicians to characterize umbilical pathologies can misdiagnose cords and is prone to error. In this talk a computational model of blood flow within rigid three-dimensional structures representative of the umbilical arteries will be presented. This study determined that the current characterization was unable to differentiate between cords which exhibited clinically distinguishable flow properties, including the cord pressure drop, which provides a measure of the loading on the fetal heart.

Sunday, November 24, 2013 2:15PM - 4:25PM –

Session D17 Biofluids: Locomotion II - Swimming 305 - Azar Eslampanah, University of Iowa

2:15PM D17.00001 Hovering of a jellyfish-like flying machine , LEIF RISTROPH, STEPHEN CHILDRESS, Courant Institute, New York University — Ornithopters, or flapping-wing aircraft, offer an alternative to helicopters in achieving maneuverability at small scales, although stabilizing such aerial vehicles remains a key challenge. Here, we present a hovering machine that achieves self-righting flight using flapping wings alone, without relying on additional aerodynamic surfaces and without feedback control. We design, construct, and test-fly a prototype that opens and closes four wings, resembling the motions of swimming jellyfish more so than any insect or bird. Lift measurements and high-speed video of free-flight are used to inform an aerodynamic model that explains the stabilization mechanism. These results show the promise of flapping-flight strategies beyond those that directly mimic the wing motions of flying animals.

2:28PM D17.00002 Do resonating bells increase jellyfish swimming performance? , ALEXANDER HOOVER, LAURA MILLER, University of North Carolina at Chapel Hill — A current question in swimming and flight is whether or not driving flexible appendages at their resonant frequency results in faster or more efficient locomotion. It has been suggested that jellyfish swim faster and/or more efficiently when the bell is driven at its resonant frequency. Previous work has modeled the jellyfish bell as a damped harmonic oscillator, and this simplified model suggests that work done by the bell is maximized when force is applied at the resonant frequency of the bell. We extend the idea of resonance phenomena of the jellyfish bell to a fluid structure interaction framework using the immersed boundary method. We first examine the effects of the bending stiffness of the bell on its resonant frequency. We then further our model with the inclusion of a "muscular" spring that connects the two sides of a 2D bell and drives it near its resonant frequency. We use this muscular spring to force the bell at varying frequencies and examine the work done by these springs and the resulting swimming speed. We finally augment our model with a flexible, passive bell margin to examine its role in propulsive efficiency.

2:41PM D17.00003 A Simple Computational Model of a jellyfish-like flying machine , FANG FANG, LEIF RISTROPH, MICHAEL SHELLEY, Applied Math Lab, Courant Institute, NYU — We explore theoretically the aerodynamics of a jellyfish-like flying machine recently fabricated at NYU. This experimental device achieves flight and hovering by opening and closing a set of flapping wings. It displays orientational flight stability without additional control surfaces or feedback control. Our model machine consists of two symmetric massless flapping wings connected to a body with mass and moment of inertia. A vortex sheet shedding and wake model is used for the flow simulation. Use of the Fast Multipole Method (FMM), and adaptive addition/deletion of vortices, allows us to simulate for long times and resolve complex wakes. We use our model to explore the physical parameters that maintain body hovering, its ascent and descent, and investigate the stability of these states.

2:54PM D17.00004 Ultra-fast Escape of a Octopus-inspired Rocket , GABRIEL WEYMOUTH[1], University of Southampton, MICHAEL TRIANTAFYLLOU[2], Massachusetts Institute of Techonology — The octopus, squid, and other cephalopods inflate with water and then release a jet to accelerate in the opposite direction. This escape mechanism is particularly interesting in the octopus because they become initially quite bluff, yet this does not hinder them in achieving impressive bursts of speed. We examine this somewhat paradoxical maneuver using a simple deflating spheroid model in both potential and viscous flow. We demonstrate that the dynamic reduction of the width of the body completely changes the flow and forces acting on the escaping rocket in three ways. First, a body which reduces in size can generate an added mass thrust which counteracts the added mass inertia. Second, the motion of the shrinking wall acts similar to suction on a static wall, reducing separation and drag forces in a viscous fluid, but that this effects depends on the rate of size change. Third, using a combination of these two features it is possible to initially load the fluid with kinetic energy when heavy and bluff and then recover that energy when streamlined and light, enabling ultra-fast accelerations. As a notable example, these mechanisms allow a shrinking spheroid rocket in a heavy inviscid fluid to achieve speeds greater than an identical rocket in the vacuum of space.

[1]Southampton Marine and Maritime Institute

[2]Center for Ocean Engineering

3:07PM D17.00005 Hydrodynamics of a Digitized Adult Humpback Whale Flipper , WESLEY N. FASSMANN, SAMUEL J. MCDONALD, SCOTT L. THOMSON, Brigham Young University, FRANK E. FISH, West Chester University — During feeding, humpback whales turn with a turn radius of up to $1/6^{th}$ of their length towards schools of fish enclosed by bubble nets. This high maneuverability requirement is facilitated by high aspect ratio flippers with leading edge tubercles that delay stall. Previous experimental and computational studies have used idealized models, such as airfoils with scalloped leading edges, to explore the influence of leading edge tubercles on boundary layer separation, vortex generation, and airfoil lift and drag characteristics. Owing to the substantial size of the flipper, no studies have been performed on a digitized adult humpback flipper with real geometry. In this study the hydrodynamics of a realistic humpback flipper model were explored. The model was developed by digitizing a sequence of 18 images circumscribing the suspended flipper of a beached humpback whale. A physical prototype was constructed based on the resulting 3D model, along with a complementary model with the tubercles removed. Experimentally-obtained measurements of lift and drag were used to study the influence of the tubercles. In the presentation, digitization and flow measurement methods are described, and the flow data and results are presented and discussed.

3:20PM D17.00006 Effects of leading edge tubercles on the flow over a humpback whale flipper[1]

, HEESU KIM, JOOHA KIM, HAECHEON CHOI, Seoul National University — In the present study, we conduct a laboratory experiment for the effect of tubercles on the hydrodynamic performance of a humpback whale flipper. The shape of the flipper used is the same as that of Miklosovic et al. (2004, 2007), and the Reynolds number considered is 100,000 based on the free-stream velocity and mean chord length. The lift and drag forces on the flipper with and without tubercles are measured by varying the angle of attack, and PIV measurements are conducted in several cross-flow planes at a few different angles of attack. As observed in previous studies, the stall angle is delayed and the maximum lift coefficient is increased. Without tubercles, the cross flow above the flipper does not show large-scale vortical motions except tip vortex. With tubercles, however, strong streamwise vortices having negative streamwise vorticity are observed along the tubercles, but the vortices with positive streamwise vorticity are either relatively weak or unobserved. This result is very different from those found in a two-dimensional wing with tubercles with which strong counter-rotating streamwise vortex pair were observed. Those vortical motions reattach the flow on the flipper and delay the separation.

[1]Supported by the NRF Programs (NRF-2011-0028032, NRF-2012K001368)

3:33PM D17.00007 ABSTRACT WITHDRAWN –

3:46PM D17.00008 Efficiency is designed into free swimming[1]

, MEHDI SAADAT, HOSSEIN HAJ-HARIRI, University of Virginia — In free swimming the swim speed and Strouhal number (St) are outputs. St alone is insufficient to decide optimal motion because many inefficient combinations of amplitude and frequency lead to the same St. This is manifested by the coincidence of the iso-lines for speed, St, and thrust. For a given combination of propulsor and body, St of motion is essentially independent of amplitude, frequency, and speed, and is only a function of shape. Some motions are efficient, and some are not. But they all have the same St. For a simple swimmer, there is a sweet spot in the dimensionless amplitude vs. frequency plane (for a fixed U) where the power efficiency is maximized. That is the place where the swimmer lives. And as long as the swimmer modulates its speed by keeping its amplitude fixed, and modulating the frequency, then the animal will always swim efficiently. So nature is efficient not because the animals are monitoring their motion in real time, but because the design of the animal is such that it cannot be inefficient.

[1]supported by ONR MURI

3:59PM D17.00009 Hydrodynamic flows can induce selective advantages among species

, FRANCESCA TESSER, Department of Physics, Eindhoven University of Technology, The Netherlands, ROBERTO BENZI, Dipartimento di Fisica, Universita' di Roma "Tor Vergata" and INFN, Roma, Italy, HERMAN J.H. CLERCX, Department of Physics, and J.M. Burgerscentrum, Eindhoven University of Technology, The Netherlands, DAVID R. NELSON, Lyman Laboratory of Physics, Harvard University, USA, PRASAD PERLEKAR, Centre for Interdisciplinary Sciences, TIFR, India, FEDERICO TOSCHI, Department of Physics, and J.M. Burgerscentrum, Eindhoven University of Technology, The Netherlands — Evolutionary forces such as genetic drift, selection, mutation and spatial diffusion act to change the genetic composition of populations. Such problems can be modeled as a system of binary reactions between competing individuals, involving births and deaths, and progressing at specific rates. An inhomogeneous or time-dependent spatial structure has the effect of modulating the interaction between individuals. To explore this problem further, we consider the dynamics and evolution of genetically diverse populations in a fluid environment where a flow field transports individuals in combination with birth and death processes [1], thus driving genetic inhomogeneities. An individual-based model in continuous space with spatial diffusion implements stochastic demographic rules for a fluctuating population size and introduces the advection of simple realistic flow fields. The system is analyzed in terms of fixation probabilities and fixation times as well as the behavior of spatial correlations. Provided organismic reproduction times are faster than the characteristic time scales of the flow, fluid ecosystems can by themselves induce spatially non-homogeneous selective advantages.

[1] Pigolotti et al. Theoretical Population Biology 84, 72 (2013)

4:12PM D17.00010 Mechanism of maximum thrust generation by oscillating compliant caudal-fin model in a quiescent fluid[1]

, HYUNGMIN PARK, YONG-JAI PARK, KYU-JIN CHO, HAECHEON CHOI, Seoul National University — A certain level of flexibility of moving appendage like a fin enhances its hydrodynamic performance (e.g., thrust generation). However, little efforts have been spent to characterize the condition in which the beneficial impact of compliance is maximized. Recent report (Park *et al.* 2012, TRO) has shown that a sinusoidally oscillating caudal-fin model generates the maximum thrust when its compliance creates a phase difference (ξ) of $\pi/2$ between the oscillating and fin-bending angles, irrespective of its planform shape. To establish the underlying mechanism, we have analyzed the time-averaged and instantaneous flow fields around nine (9) oscillating caudal-fin models with varying their compliance. A series of particle image velocimetry measurements were performed in a quiescent water tank. When $\xi < \pi/2$, a strong interaction between the separated trailing-edge vortex (TEV) and the TEV forming at next stroke directs the flow in a transverse way, thereby enhancing the decay of thrust-generating jet velocity. At $\xi = \pi/2$, this interaction is weak such that the fast jet velocity is retained along the streamwise direction. On the other hand, when $\xi > \pi/2$, the trailing-edge is moving opposite to the oscillation reducing the rotational circulation of TEV.

[1]Supported by the NRF programs (NRF-2011-0028032, NRF-2013R1A1A1008373, NRF-2012M2A8A4055647), Korea.

Sunday, November 24, 2013 2:15PM - 4:25PM –

Session D18 Biofluids: General II - Collective Behavior and Microswimmers 306/307 - Alireza Karimi, University of Notre Dame

2:15PM D18.00001 Impact of Viscoelasticity on the Coordinated Swimming of Motile Bacteria

, ALIREZA KARIMI, AREZOO ARDEKANI, University of Notre Dame — The formation of bacterial communities is often associated with production of extracellular polymeric substances which impart viscoelastic behavior to the surrounding fluid. This phenomenon greatly affects the hydrodynamic interactions of swimming bacteria and the resulting chaotic dynamics. To investigate this process, we used a kinetic model developed to study the behavior of self-propelled particles in conjunction with Oldroyd-B constitutive equation and the Stokes equations. Using large-scale numerical simulations of the system, we analyzed the effect of the viscoelasticity on the coordinated behavior of the microorganisms. In addition, by varying the corresponding parameters of the problem such as Weissenberg number and viscosity ratio, we explored different flow regimes in order to gain insight regarding the characteristics of the flow patterns induced by the collective motion of motile bacteria.

2:28PM D18.00002 Diffusion of passive particles in active suspensions , MATTHIAS MUSSLER, Saarland University, SALIMA RAFAI, UJF Grenoble, Liphy, THOMAS JOHN, Saarland University, PHILIPPE PEYLA, UJF Grenoble, Liphy, CHRISTIAN WAGNER, Saarland University — We study how an active suspension consisting of a definite volume fraction of the microswimmer *Chlamydomonas Reinhardtii* modifies the Brownian movement of small to medium size microspheres. We present measurements and simulations of trajectories of microspheres with a diameter of 20 μm in suspensions of *Chlamydomonas Reinhardtii*, a so called "puller," and show that the mean squared displacement of such trajectories consist of parabolic and a linear part. The linear part is due to the hydrodynamic noise of the microswimmers while the parabolic part is a consequence of directed motion events that occur randomly, when a microsphere is transported by a microswimmer on a timescale that is in higher order of magnitude than the Brownian like hydrodynamic interaction. In addition, we theoretically describe this effect with a dimensional analysis that takes the force dipole model used to describe "puller" like *Chlamydomonas Reinhardtii* into account.

2:41PM D18.00003 A drag-based mechanism for vertical force production in the smallest flying insects , SHANNON JONES, RYAN LAURENZA, LAURA MILLER, UNC-Chapel Hill — Previous work has shown that the flight kinematics and aerodynamics of the smallest flying insects may be significantly different than that of their larger counterparts. These small insects, such as thrips and parasitoid wasps, are on the order of 1 mm in length and operate at a Reynolds number less than 10. Due to their small size and high wing beat frequency, quantitative data on the wing kinematics of the smallest insects is not available. As a result, there has been much debate and speculation about the flight strategies employed by these insects. With the challenges associated with generating lift at low Reynolds numbers, it could be beneficial for the smallest insects to use a drag-based motion to generate some or all of its vertical force, however this has not been rigorously investigated. We used computational fluid dynamics to investigate the feasibility of drag-based propulsion in the tiniest insects. We investigated the vertical force generated by an idealized drag-based vertical stroke over a range of Reynolds numbers from 1 to 150. We also compared this stroke to more conventional hovering stroke kinematics such as that of a fruit fly and dragonfly.

2:54PM D18.00004 Emergent structures and dynamics in suspensions of self-phoretic colloids , ANDREA SCAGLIARINI, Department of Physics, University of Rome "Tor Vergata", IGNACIO PAGONABARRAGA, Department of Fundamental Physics, University of Barcelona — Active fluids, such as suspensions of self-propelled particles , are a fascinating example of Soft Matter displaying complex collective behaviours which provide challenges in non-equilibrium Statistical Physics. The recent development of techniques to assemble miniaturized devices has led to a growing interest for micro and nanoscale engines that can perform autonomous motion ("microrobots"), as, for instance, self-phoretic colloids, for which the propulsion is induced by the generation of a chemical species in a reaction catalyzed at the particle surface. We perform a mesoscopic numerical study of suspensions of self-phoretic colloids. We show that, at changing the sign of the phoretic mobility (which accounts for the colloid-solute interactions), the system switches from a cluster phase to a state with slowed dynamics. We find that the cluster size distribution follows an exponential behaviour, with a characteristic size growing linearly with the colloid activity, while the density fluctuations grow as a power-law with an exponent depending on the cluster fractal dimension.We single out hydrodynamic interactions, showing that their effect is to work against cluster formation. For positive μ, we observe that colloids tend to reach an ordered state on a triangular lattice.

3:07PM D18.00005 Hydrodynamics of Choanoflagellate Feeding , ANDERS ANDERSEN, Department of Physics, Technical University of Denmark and Centre for Ocean Life, Villum Kann Rasmussen Centre of Excellence, LASSE TOR NIELSEN, THOMAS KIORBOE, National Institute of Aquatic Resources, Technical University of Denmark and Centre for Ocean Life, Villum Kann Rasmussen Centre of Excellence — Choanoflagellate filter feeding is a poorly understood process. Studies indicate that the pressure differences created by the beating of the flagellum are insufficient to produce an adequate water flow through the collar filter, the mechanism believed to ultimately transport food particles to the cell. The collar is composed of numerous microvilli arranged as a palisade, and the low porosity of the filter provides high resistance to the water flow. Additionally, ultrastructural studies often show signs of mucus-like substances in and around the collar, potentially further hampering water flow. We present high-speed video of live material showing the particle retention and the beating of the flagellum in the choanoflagellate species Diaphanoeca grandis. We use the observations as input to model the low Reynolds number fluid dynamics of the fluid force produced by the flagellum and the resulting feeding flow.

3:20PM D18.00006 Localized structure of Euglena bioconvection , MAKOTO IIMA, ERIKA SHOJI, AKINORI AWAZU, HIRAKU NISHIMORI, SHUNSUKE IZUMI, Hiroshima University, HIROSHIMA UNIVERSITY COLLABORATION — Bioconvection of a suspension of Euglena gracilis, a photosensitive flagellate whose body length is approximately 50 micrometers, was experimentally studied. Under strong light intensity, Euglena has a negative phototaxis; they tend to go away from the light source. When the bright illumination is given from the bottom, a large scale spatio-temporal pattern is generated as a result of interaction between Euglena and surrounding flow. Recently, localized convection pattern had been reported, however, the generation process and interaction of the localized convection cells has not been analyzed. We performed experimental study to understand the localization mechanism, in particular, the onset of bioconvection and lateral localization behavior due to phototaxis. Experiments started from different initial condition suggests a bistability near the onset of the convection as binary fluid convection that also shows localized convection cells. Dynamics of localized convections cells, which is similar to the binary fluid convection case although the basic equations are not the same, is also reported.

3:33PM D18.00007 Limit cycle dynamics in swimming systems , CYNDEE FINKEL, KARL VON ELLENRIEDER, Florida Atlantic University — An experimental apparatus was constructed to model basic features expected in the flow about a freely swimming fish. A D-shaped cylinder is used to represent the body and an oscillating foil, the tail. The swimming system is suspended in a constant freestream flow. A closed loop PI controller is used to maintain a set point, stream-wise location. The system is released from multiple downstream and upstream locations and permitted to swim to the set point. The Strouhal number measured when the swimming system achieves a constant forward swimming speed is compared to values observed in nature. The results suggest that self-regulation passively selects the Strouhal number and that no other external sensory input is necessary for this to happen. This self-regulation is a result of a limit cycle process that stems from nonlinear periodic oscillations. Phase plane analyses are used to examine the synchronous conditions due to the coupling of the foil and wake vortices. It is shown that the phase locking indices depend on the Strouhal number and approach a frequency locking ratio of about 0.5. The results suggest that Strouhal number selection in steady forward natural swimming is the result of a limit cycle process and not actively controlled by an organism.

3:46PM D18.00008 Local fluid transport by planktonic swarms , MONICA MARTINEZ-ORTIZ, California Institute of Technology, JOHN DABIRI, Graduate Aeronautical Laboratories and Bioengineering, California Institute of Technology — Energy transport in the ocean occurs through an intricate set of pathways mainly powered by physical phenomena. The hypothesis that vertical migrations of aquatic fauna may contribute to this process through the action of the induced drift mechanism has been investigated in recent years. Microscale measurements by Kunze et al (1), in Saanich Inlet have shown the presence of high kinetic energy dissipation rates in the vicinity of vertically migrating krill swarms. However, it remains uncertain if energy is being introduced at scales large enough to induce the transport of fluid across surfaces of equal density. Within this context, the present study aims to provide experimental insight of fluid transport by planktonic swarms. The vertical migration of Artemia salina is triggered and controlled by means of a system of stationary and translating luminescent signals. High speed flow visualizations elucidate the competing effects of upward drift by the passive sections of the organisms and downward flow induced by the appendages. The resulting fluid transport is assessed by using PIV at different stages of the migration. The kinetic energy spectrum is computed using velocity correlation functions to determine the length scales at which the animals introduce energy to the flow.

3:59PM D18.00009 Why do mayflies switch from rowing to flapping as they grow?[1] , RODOLPHE CHABREYRIE, Department of Mechanical and Aerospace Engineering, The George Washington University, KHALED ABDELAZIZ, Department of Mechanical Engineering, University of Maryland, ELIAS BALARAS, Department of Mechanical and Aerospace Engineering, The George Washington University, KEN KIGER, Department of Mechanical Engineering, University of Maryland — In order to maintain its metabolism, many species of mayfly nymph utilizes an oscillating array of wing-shaped gills to augment its extraction of dissolved oxygen from the surrounding water. As the nymph develops, the kinematics of these gills are observed to abruptly change from a rowing-like to flapping-like motion. In order to understand the role of this abrupt kinematic change, we consider a pure Lagrangian approach, looking at the mayfly as a stirring device. Using this Lagrangian approach we are able to provide the reason behind the observed kinematic transition during ontogeny. More precisely, recent and powerful tools from chaos theory are applied to in-sillico mayfly nymph experiments. In this talk, we show both qualitatively and quantitatively how the change of kinematics enables a better attraction, stirring and confinement of dissolved oxygen charged water within the near proximity of the gills surface. From the computational velocity field we reveal attracting barriers to transport, i.e. attracting Lagrangian coherent structures (LCS), that form the transport skeleton between and around the gills. In addition, we quantify how well advected particles and consequently dissolved oxygen is spread and mixed within the gills region.

[1]We acknowledge the support of NSF award CBET-1067066.

4:12PM D18.00010 Simulation of collective behaviour in micro-scale swimmers: Effects of tumbling and rotary diffusion , DEEPAK KRISHNAMURTHY, GANESH SUBRAMANIAN, Jawaharlal Nehru Centre for Advanced Scientific Research — Recent experiments have shown that suspensions of swimming micro-organisms are characterized by complex dynamics involving enhanced swimming speeds, large-scale correlated motions and enhanced tracer diffusion. Understanding this dynamics is of fundamental interest and also has relevance to biological systems. In this work we develop a particle-based computational model to study a suspension of hydrodynamically interacting rod-like swimmers with the relation between the swimming velocity and intrinsic stress being enforced from slender body theory. Such an *a priori* specification reduces the computational cost since one now has a "kinematic" simulation with a fixed interaction law between swimmers; this does not restrict our study of the dynamics since the destabilizing mechanism has been attributed to the intrinsic (rather than the induced) stress field. Importantly, the model will include intrinsic de-correlation mechanisms found in bacteria such as rotary diffusion and tumbling whose effects have so far not been studied via simulations. Using this model we predict a box-size independent stability threshold based on the suspension concentration, tumble-time (duration between subsequent tumble events) and rotary diffusivity. Comparisons are made with the linear stability theory predictions by Subramanian & Koch (JFM 2009). We demonstrate that the effect of tumbling and rotary diffusion is to stabilize the suspension.

Sunday, November 24, 2013 2:15PM - 4:25PM –

Session D19 Biofluids: Cellular I - Computational Studies on Cellular Kinematics 310/311 - Timothy Wei, University of Nebraska, Lincoln

2:15PM D19.00001 Fluid Flow in Cell Printing , MAZIYAR JALAAL, Department of Mechanical Engineering, The University of British Columbia, Vancouver, BC, Canada, ERIC CHENG, ALI AHMADI, KAREN CHEUNG, BORIS STOEBER, Department of Electrical and Computer Engineering, The University of British Columbia, Vancouver, BC, Canada — Inkjet drop-on-demand (DOD) dispensing of cells has numerous applications including cell-based assays and tissue engineering. In our experiments, using a transparent inkjet nozzle, high speed camera, and a shadowgraphy technique, we have observed three different characteristic cell behaviors during droplet ejection: 1) traveling toward the nozzle tip, 2) ejection from the nozzle, and 3) reflection away from the nozzle tip, where the reflection is an unwanted effect which contributes to the unpredictability of current cell printing systems. To understand the reflection mechanisms, we use numerical simulation to resolve the fluid motion inside the nozzle in presence of a cell during drop formation. For this purpose an adaptive finite volume method is employed. To track the interfaces (cell-liquid, gas-liquid) a volume of fluid (VOF) method is used, where the cell is modeled as an immiscible fluid droplet with different physical properties from the suspending fluid. It is shown that after a short period of time, a recirculation zone close to the nozzle tip is generated due to droplet pinch-off. This causes a reverse flow (velocity away from the nozzle) in the center of the nozzle. This dynamic flow field inside the nozzle causes a cell to show one of the three behaviors described above depending on its initial position. Moreover, it is shown that, depending on the size, deformability, and location of the cell, the drop formation process may be influenced.

2:28PM D19.00002 Numerical computations of ionic electrodiffusion and osmotic water flow in cells , LINGXING YAO, Case Western Reserve University, YOICHIRO MORI, University of Minnesota — We develop a computational method to characterize ionic electrodiffusion and osmotic water flow in cellular systems. In the biological model system we used, cell membranes, which are permeable to both water and ionic flows, divide the domain into intracellular and extracellular regions. The cell membranes move with the flow it is embedded in, while its elastic force and osmotic forces due to ions will in turn affect fluid properties. The whole system is then consists of fluid-structure interactions, coupled with ionic electrodiffusion on domain with moving (internal) interfaces. The numerical computation of advection-diffusion in a 2d rectangle domain with moving boundaries is carried out by using a embedded Cartesian grid method over the entire rectangle domain, which represents the intra- and extracellular regions, while the fluid-structure interactions is handled by the Immersed Boundary Method. We will describe our numerical scheme of solving this PDE system and illustrate the results with some simple applications as the proof of principles.

2:41PM D19.00003 Numerical Modeling of Flow through Phloem Considering Active Loading[1] , JIN LIU, TSUN-KAY JACKIE SZE, PRASHANTA DUTTA, Washington State University, Pullman, WA — Transport through phloem is of significant interest in engineering applications including self-powered microfluidic pumps. We present a phloem model, combining protein level mechanics with cellular level fluid transport. Fluid flow and sucrose transport through a petiole sieve tube are simulated using the Nernst-Planck, Navier-Stokes, and continuity equations. Governing equations are solved using the finite volume method with dynamically calculated boundary conditions. Sieve tube cell structure consisting of sieve plates is included in a two dimensional model by computational cell blocking. Sucrose transport is incorporated as a boundary condition through a six-state model, bringing in active loading mechanisms with consideration of physical plant properties. The effects of reaction rates and leaf sucrose concentration are investigated to understand the transport mechanism in petiole sieve tubes. Numerical results show that increasing forward reactions of the proton sucrose transporter significantly promotes the pumping ability. A lower leaf sieve sucrose concentration results in a lower wall inflow velocity, but yields a higher inflow of water due to the active loading mechanism. The overall effect is higher outflow velocity for lower leaf sieve sucrose concentration because the increase in inflow velocity outweighs wall velocity. This new phloem model provides new insights on mechanisms potentially useful for fluidic pumping in self-powered microfluidic pumps.

[1]This work is supported in part by the National Science Fundation grant CBET-1250107

2:54PM D19.00004 The Effect of Shape Memory on Red Blood Cell Motions , XITING NIU, LINGLING SHI, TSORNG-WHAY PAN, ROLAND GLOWINSKI, University of Houston — An elastic spring model is applied to study the effect of the shape memory on the motion of red blood cell in flows. In shear flow, shape memory also plays an important role to obtain all three motions: tumbling, swinging, and tank-treading. In Poiseuille flow, cell has an equilibrium shape as a slipper or parachute depending on capillary number. To ensure the tank-treading motion while in slippery shape, a modified model is proposed by introducing a shape memory coefficient which describes the degree of shape memory in cells. The effect of the coefficient on the cell motion of red blood cell will be presented.

3:07PM D19.00005 Numerical simulation of red blood cell suspensions behind a moving interface in a capillary[1] , SHIHAI ZHAO, TSORNG-WHAY PAN, Department of Mathematics, University of Houston, Houston, TX, USA — Computational modeling and simulation are presented on the motion of red blood cells behind a moving interface in a capillary. The methodology is based on an immersed boundary method and the skeleton structure of the red blood cell (RBC) membrane is modeled as a spring network. The computational domain is moving with either a designated RBC or an interface in an infinitely long two-dimensional channel with an undisturbed flow field in front of the domain. The tanking-treading and the inclination angle of a cell in a simple shear flow are briefly discussed for the validation purpose. We then present the results of the motion of red blood cells behind a moving interface in a capillary, which show that the RBCs with higher velocity than the interface speed form a concentrated slug behind the interface. It is a key mechanism responsible for penetration failure in a capillary behind the meniscus.

[1]This work is funded by NSF.

3:20PM D19.00006 Probing bilayer–cytoskeletal interactions in erythrocytes using a two-component dissipative particle dynamics model[1] , ZHANGLI PENG, Massachusetts Institute of Technology, XUEJIN LI, Brown University, IGOR PIVKIN, University of Lugano, MING DAO, Massachusetts Institute of Technology, GEORGE KARNIADAKIS, Brown University — We develop a two-component dissipative particle dynamics (DPD) model of the red blood cell (RBC) membrane by modeling the lipid bilayer and the cytoskeleton separately. By applying this model to simulate four different experiments on RBCs, including micropipette aspiration, membrane fluctuations, tank-treading motions in shear flow and bilayer tethering in a flow channel, we validated our model and studied the mechanical properties of the bilayer–cytoskeletal interaction in a systematic and controlled manner, such as its elastic stiffness, viscous friction and strength. In the same time, we also resolved several controversies in RBC mechanics, e.g., the dependence of tank-treading frequency on shear rates and the possibility of bilayer–cytoskeletal slip. Furthermore, to investigate RBC dynamics in the microcirculation, we simulated the passages of RBCs through narrow channels of the flow cytometer in vitro and their passages through the splenic inter-endothelial slits in vivo. The effects of RBC geometry and membrane stiffness on the critical pressure gradient of passage were studied, and the simulation results agree well with experimental measurements.

[1]This work was supported by National Institutes of Health Grant R01HL094270 and the new Department of Energy Collaboratory on Mathematics for Mesoscopic Modeling of Materials (CM4).

3:33PM D19.00007 Quantifying the transition of blood flow to the non-continuum regime , HUAN LEI, Pacific northwest national lab, DMITRY FEDOSOV, Institute of Complex Systems (ICS-2), Research Center Juelich, BRUCE CASWELL, GEORGE KARNIADAKIS, Brown University — Blood flow is usually treated as a Newtonian fluid down to diameters of about $200\ \mu m$. We employ the dissipative particle dynamics to simulate the flow of red blood cell suspensions driven through small tubes (diameters $10 - 150\ \mu m$) in the range marking the transition from venules to the large capillaries. Simulation results show that for diameters less than about $100\ \mu m$ the suspension's stress cannot be described as a continuum, even a heterogeneous one. In tube flow the cross-stream stress gradient induces an inhomogeneous distribution of RBCs featuring a centerline cell density peak, and a cell-free layer next to the wall. The local viscosity across the section as a function of the strain rate is found to be essentially independent of tube size for the larger diameters and is determined by the local hematocrit (H) and shear rate. As the tube diameter decreases below about $100\ \mu m$, the viscosity in the central region departs from the large-tube similarity function of the shear rate, since H increases significantly towards the centerline. The dependence of shear stress on tube size, in addition to the expected local shear rate and local hematocrit, implies that blood flow in small tubes cannot be described as a heterogeneous continuum.

3:46PM D19.00008 2-Point Particle Tracking Microrheology of Directional Gels , MANUEL GOMEZ-GONZALEZ, JUAN C. DEL ALAMO, University of California, San Diego — The stiffness of the cell cytoplasm, and other minute-quantity materials, can be measured by using Particle Tracking Microrheology, where a micron size spherical particle is used as a probe. It relies on the assumption of isotropy of the probed material. In order to apply it to highly oriented materials we have calculated the drag force of a microparticle embedded in a directional viscoelastic gel. The gel is modeled as a directional viscoelastic network frictionally coupled to a viscous isotropic fluid. The directional network is modeled with the Leslie-Ericksen equations and the isotropic fluid with the Stokes equation. The motion of particles embedded in such a directional gel is dependent on up to three viscoelastic coefficients, but only two can be calculated from tracking a single probing particle. We have calculated the first order perturbation that the motion of one probe induces on a distant particle, as a function of the three viscosity coefficients. By correlating the motion of two distant particles we can measure such a perturbation and obtain three independent equations that univocally determine the three viscoelasticity coefficients that define a directional viscoelastic gel.

3:59PM D19.00009 Ordered and chaotic flow of red blood cells flowing in a narrow tube[1] , NATALIE N. BEAMS, JONATHAN B. 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, small perturbations from the center of the tube can cause this order to break down, resulting in apparently chaotic flow. Investigating this breakdown using a high-fidelity boundary integral solver for flowing blood cells, we show the existence of a bifurcation point for the appearance of this non-orderly behavior. The bifurcation point is found to be dependent on the diameter of the tube and the density of the cells, with more cells required to instigate chaotic behavior in smaller tubes (e.g., 27.5% cells by volume for a 11.28μm diameter tube, but only 5.63% for similar behavior in a tube twice that width). Increasing the cell interior viscosity is also observed to increase the amplification of perturbations. Additionally, as a counterpoint, we show that cells flowing chaotically in $D = 12\mu$m tubes, apparently indefinitely, will slowly organize into a regular single file if D is decreased to $D \approx 10\mu$m.

[1]Supported by NSF

4:12PM D19.00010 Three-dimensional simulations of the cell growth and cytokinesis using an immersed boundary method[1] , YIBAO LI, JUNG-IL CHOI, Dept. CSE, Yonsei University — For an animal cell, cytokinesis is the process by which a cell divides its cytoplasm to produce two daughter cells. We present a three-dimensional immersed boundary method for the simulation of cell growth and cytokinesis. The proposed model is robust and realistic in deciding the position of the cleavage furrow and in defining the contractile force leading to cell division. For accurate calculations, a simple surface re-meshing algorithm is applied to uniformalize distorted meshes. In addition, to keep the mass conservation of the numerical solution at each time step, we use the volume-preserving scheme (Li et al., 2013). We investigate the effects of each model parameter on the cell growth and cytokinesis, and compare numerical results with the experimental data to demonstrate the efficiency and accuracy of the proposed method.

[1]Supported by WCU program (R31-10049) of NRF.

Sunday, November 24, 2013 2:15PM - 4:12PM –

Session D20 Boundary Layers III: Flow over Roughness Elements 315 - Luciano Castillo, Texas Tech University

2:15PM D20.00001 Time Resolved Tomographic PIV Measurements of Rough-Wall Turbulent Channel Flow[1] , RINALDO MIORINI, CAO ZHANG, JOSEPH KATZ, Johns Hopkins University — Time resolved tomographic PIV is used to study flow structures in the outer region of a rough-wall turbulent boundary layer, focusing on imprints of the roughness on the outer layer. Measurements are performed in a transparent channel installed in the JHU optically index matched facility. The roughness consists of pyramids with height, k=0.46 mm, and wavelength, λ =3.2 mm, satisfying h/k=55 (h=25.4 mm is the channel half-height), k^+ =64 and Re=40000. The TPIV setup consists of four high-speed cameras operating at 3 kHz, which view the sample volume through acrylic prisms. The flow field is illuminated by an Nd:YLF laser. Following enhancement, calibration, and reconstruction, 64^3 voxels interrogation volumes with 0.75 overlap provide 3D velocity fields with spacing of 0.588^3 mm^3. Formation and transport of near-wall 3D U-shaped vortex structures, with base in front of the pyramids, and quasi-streamwise legs extending between pyramid crest lines are evident from the data. Extended streamwise regions of high wall-normal vorticity appear "latched" to the roughness elements close to the wall, but are transported downstream at higher elevations. Also evident are traveling streamwise low velocity streaks, which cover many roughness elements.

[1]Sponsored by NSF CBET and ONR.

2:28PM D20.00002 The rough-wall turbulent boundary layer revisited , P. HENRIK ALFREDSSON, ANTONIO SEGALINI, Linné FLOW Centre, KTH, IAN P. CASTRO, University of Southampton — The turbulent boundary layer over a rough wall is a canonical flow case that, despite its long history, is still a subject of numerous debates regarding its appropriate scaling and the physical phenomena involved. In a recent paper[1] a novel empirical scaling, based on the so called diagnostic plot, has been proposed to describe the streamwise velocity variance profile for rough-wall conditions. The scaling gives the turbulence intensity as a function of the local mean velocity and of the roughness function, ΔU^+, regardless of the specific roughness morphology. The analysis is reviewed and extended to other components of the stress tensor, with the aim of providing a simple description of the turbulence statistics over rough surfaces and to establish a connection between rough and smooth surfaces beyond Townsend's hypothesis. Several independent experiments and numerical simulations are used for this purpose. Such an estimation of the turbulence statistical characteristics will be useful for model validation and for a variety of applications that range from wind energy to micrometeorology.

[1]Castro, I.P., Segalini, A. & Alfredsson, P.H. 2013 Outer-layer turbulence intensities in smooth- and rough-wall boundary layers, J. Fluid Mech. **727**, 119–13

2:41PM D20.00003 Atmospheric boundary layer flow over transverse roughness transitions: induced mixing and flow characterization , DAVID WILLINGHAM, WILLIAM ANDERSON, Baylor University — The response of turbulent atmospheric boundary layer flow to abrupt surface roughness heterogeneities has been discussed extensively in previous literature. However, many prior studies exclusively consider cases in which the streamwise flow is aligned perpendicular to the roughness heterogeneity, representing the noted smooth-to-rough or rough-to-smooth transition. This work seeks to identify the affects of parallel-aligned roughness transitions on turbulent boundary layer flow and determine whether these effects are non-negligible. To this end, a set of large eddy simulations have been performed over surfaces composed of parallel strips of high roughness adjacent to low roughness. The width of these strips, as well as the ratio of high to low roughness lengths were systematically varied between simulations. Close to the surface, there is a transverse gradient in streamwise velocity owing to the differing roughness lengths, and this gradient induces transverse mixing which serves to introduce an important secondary flow in the boundary layer. Low and high momentum pathways (Mejia-Alvarez et al.) are observed to form in the upper region of the boundary layer above the low and high roughness regions, respectively. Associated with this is the formation of boundary layer-scale counter-rotating vortices, adjacent to the high roughness strips. Interestingly, we find that even modest differences between the high and low roughness length is adequate to induce this process.

2:54PM D20.00004 Numerical simulations of flow over realistic rough surfaces , JUNLIN YUAN, UGO PIOMELLI, Queen's University — Large-eddy simulations are carried out on open-channel flows over multiple types of rough surfaces occurring in hydraulic turbine in both transitionally and fully rough regimes with the Kármán number ranging from 400 to 1000. The roughness imposed using an immersed boundary method is spatially resolved by the grid. The roughness functions are used to test several correlations proposed in the literature to relate surface parameters to the equivalent sand-grain height; agreement is obtained with experimental results on gas turbine roughness, despite slight differences in model coefficients. For relatively sparse distributions, the realistic roughness yields a higher drag compared to modeled roughness. The mean-flow ejecting and sweeping motions as part of the channeling phenomenon contribute to vertical momentum transports and correlate closely to regions of positive surface slope. It is observed that a stronger mean flow effect corresponds to higher frequency of relatively strong bursting events in the near-wall region, while the average size of these events is controlled by roughness length scales that are separated from the ones determining the event frequency. Further discussions will be given on possible indications of important surface parameters.

3:07PM D20.00005 Characteristics of Larger-Scale Motions in Turbulent Flow Overlying Multi-Scale Roughness , J.M. BARROS, K.T. CHRISTENSEN, Univ. of Illinois — The energy and Reynolds-stress content of large-scale motions (LSM) and superstructures within turbulent flow overlying a multi-scale roughness topography are explored using high-frame-rate stereo particle image velocimetry 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 arranged in a highly irregular manner. Previous measurements revealed that this roughness induces significant spanwise heterogeneity in the turbulence statistics in the form of low- and high-momentum flow pathways marked by enhanced Reynolds stresses and turbulent kinetic energy. Frequency spectra of streamwise velocity at fixed wall-normal location also display strong dependence on spanwise position. In particular, the roughness promotes enhanced energy content of the LSMs and smaller-scale motions. Depending on spanwise location, pre-multiplied spectra highlight significant modification of the energy content of the superstructures due to roughness compared to smooth-wall flow.

3:20PM D20.00006 ABSTRACT WITHDRAWN –

3:33PM D20.00007 Parametric analysis of turbulent wall jet in still air over a transitional rough surface: Universal relations , NOOR AFZAL, Retired — The novel scalings for streamwise variations of the flow in a turbulent wall jet over a fully smooth, transitional and fully rough surfaces have been analyzed. The universal scaling for arbitrary wall roughness is condidered in terms of the roughness friction Reynolds number (that arises from the stream wise variations of roughness in the flow direction) and roughness Reynolds number at the nozzle jet exit. The transitional rough wall jet functional forms have been proposed, whose numerical constants power law index and prefactor are estimated from best fit to the data for several variables, like, maximum wall jet velocity, boundary layer thickness at maxima of wall jet velocity, the jet half width, the friction factor and momentum integral, which are supported by the experimental data. The data shows that the two asymptotes of fully rough and fully smooth surfaces are co-linear with transitional rough surface, predicting same constants for any variable of flow for full smooth, fully rough and transitional rough surfaces. There is no universality of scalings in terms of traditional variables as different expressions are needed for each stage of the transitional roughness. The experimental data provides very good support to our universal relations.

3:46PM D20.00008 Direct numerical simulation of a turbulent rough-walled pipe , ANDREW OOI, LEON CHAN, MICHAEL MACDONALD, NICHOLAS HUTCHINS, DANIEL CHUNG, University of Melbourne — Direct Numerical Simulations (DNS) of turbulent pipe flow have been conducted at low to medium Reynolds numbers. The surface of the pipe is varied from a smooth-wall pipe to a rough-wall pipe, where the roughness is comprised of three-dimensional sinusoidal elements. Parametric tests were carried out to analyse the effects of the height and the wavelength of the sinusoidal surface. An analysis of the mean statistics convincingly supports Townsend's outer-layer hypothesis. Higher-order statistics such as skewness and flatness are also gathered and show reasonable collapse in the outer layer of the pipe for different roughness cases. Even at a mean-to-peak roughness height of 2.5 wall units, which is within the viscous sublayer of the pipe, the roughness effected a centerline velocity shift of 0.50 friction velocities. When the roughness height is increased to a mean-to-peak roughness height of 20 wall units, the flow can be considered to be in the fully rough regime, with a centerline velocity shift of 7.1 friction velocities.

3:59PM D20.00009 Perturbation of turbulent channel flow structure by a cylindrical roughness element , G. PATHIKONDA, K.T. CHRISTENSEN, Univ. of Illinois — The existence and dynamic significance of coherent structures in turbulent wall-bounded flows has led to substantial interest in both understanding and perhaps manipulating these structures. To this end, the current work investigates the perturbing influence of a single, wall-mounted cylindrical element in the log layer of a fully-developed turbulent channel flow ($\mathrm{Re}_\tau \sim 1250$), and the latter's response and subsequent evolution. Hot-wire measurements were made in the wake of these elements (~ 250 viscous wall units tall; approximately 20% of the channel half-height) at various streamwise positions. Mean and spectral statistics of the fluctuating streamwise velocity indicate preferential and inhomogeneous modifications to the incident flow by the cylinder. Subsequent recovery in pre-multiplied spectra of the perturbed flow downstream suggests a biased environment conductive to structures corresponding to the very-large-scale motions (superstructures). Though in an inhomogeneous manner, the flow was found to continuously recover towards the unperturbed incident flow with increasing downstream distance.

Sunday, November 24, 2013 2:15PM - 4:25PM –

Session D21 Turbulence: Simulations II - DNS and LES I 316 - Krishnan Mahesh, University of Minnesota

2:15PM D21.00001 Towards Feature-Resolved Simulations of Superhydrophobic Surfaces[1] , YIXUAN LI, KARIM ALAME, KRISHNAN MAHESH, University of Minnesota — Superhydrophobic surfaces have potential for viscous friction reduction, anti-corrosive protective coatings and self-cleaning techniques. Most previous studies focused on large scale grooves or pillars in the laminar regime. In this study, two fully covered microtextured superhydrophobic surfaces and two unit microtextured surfaces with different geometries (grooves and posts) are tested in both laminar and turbulent flows using DNS. Slip length and discharge are computed in the laminar regime and compared with theoretical estimate and experiment. The turbulent simulations are performed for both "unit cells" as well as the entire textured surfaces. Fully wetted simulations reveal the effect that geometry alone exerts. A volume of fluid methodology is being developed towards allowing for air/water interfaces inside the grooves, and will be discussed.

[1]Supported by Office of Naval Research.

2:28PM D21.00002 Direct numerical simulation from laminar to fully-developed turbulence in spatially evolving pipe flow and flat plate boundary layer , XIAOHUA WU, Royal Military College of Canada, PARVIZ MOIN, Center for Turbulence Research, Stanford University, RONALD J. ADRIAN, JON R. BALTZER, Arizona State University, JEAN-PIERRE HICKEY, Center for Turbulence Research, Stanford University — Direct numerical simulations of spatially evolving pipe flow and boundary layer have been performed. The pipe is 250R long, the flow Reynolds number is 6000 and 8000, and the calculation used up to 1.7 billion grid points. Pipe inlet disturbance is from a very-thin wire ring placed at different radial locations. It is found that energy norm in the flow downstream of such disturbance can grow exponentially with axial distance. The boundary layer's momentum thickness Reynolds number develops from 80 to 3000 with a free-stream turbulence intensity decaying from 3 percent to 0.8 percent. Its mesh has 4 billion grid points. Good quantitative agreement with experimental data is obtained. In both the pipe flow and the boundary layer, under these inlet disturbances, Lambda vortex, hairpin packet, infant turbulent spot, mature turbulent spot, and hairpin forest occur naturally and sequentially. Passive scalar was also introduced in the simulation in a manner analogous to the color band experiment of Osborne Reynolds.

2:41PM D21.00003 Application of the High Gradient hydrodynamics code to simulations of a two-dimensional zero-pressure-gradient turbulent boundary layer over a flat plate[1] , BRYAN E. KAISER, SVETLANA V. POROSEVA, University of New Mexico, JESSE M. CANFIELD, JEREMY A. SAUER, Geophysical Fluid Dynamics Institute, Florida State University, RODMAN R. LINN, Los Alamos National Laboratory — The High Gradient hydrodynamics (HIGRAD) code is an atmospheric computational fluid dynamics code created by Los Alamos National Laboratory to accurately represent flows characterized by sharp gradients in velocity, concentration, and temperature. HIGRAD uses a fully compressible finite-volume formulation for explicit Large Eddy Simulation (LES) and features an advection scheme that is second-order accurate in time and space. In the current study, boundary conditions implemented in HIGRAD are varied to find those that better reproduce the reduced physics of a flat plate boundary layer to compare with complex physics of the atmospheric boundary layer. Numerical predictions are compared with available DNS, experimental, and LES data obtained by other researchers. High-order turbulence statistics are collected. The Reynolds number based on the free-stream velocity and the momentum thickness is 120 at the inflow and the Mach number for the flow is 0.2. Results are compared at Reynolds numbers of 670 and 1410.

[1]A part of the material is based upon work supported by NASA under award NNX12AJ61A and by the Junior Faculty UNM-LANL Collaborative Research Grant.

2:54PM D21.00004 Estimating the Effective Reynolds Number in Implicit Large Eddy Simulation , FERNANDO GRINSTEIN, LANL, YE ZHOU, LLNL, ADAM WACHTOR, BRIAN HAINES, LANL — In implicit large-eddy simulation (ILES) energy-containing large scales are resolved, and physics capturing numerics are used to spatially filter-out unresolved scales and implicitly model subgrid scale effects. From an applied perspective, it is highly desirable to estimate a characteristic Reynolds number (Re) – and therefore a relevant effective viscosity, so that the impact of resolution on predicted flow quantities and their macroscopic convergence can be usefully characterized. We argue in favor of obtaining robust Re estimates away from the smallest scales of the simulated flow – where numerically controlled dissipation takes place, and propose theoretical basis and framework to determine such measures. ILES examples include forced turbulence as a steady flow case, the Taylor-Green vortex to address transition and decaying turbulence, and simulations of a laser-driven reshock experiment illustrating a fairly complex turbulence problem of current practical interest.

3:07PM D21.00005 Large-Eddy Simulation of Propeller Crashback[1] , PRAVEEN KUMAR, 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 propeller crashback. Results are shown for a stand-alone open propeller, hull-attached open propeller and a ducted propeller. The simulations are compared to experiment, and used to discuss the essential physics behind the unsteady loads.

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

3:20PM D21.00006 Large Eddy Simulation of Entropy Generation in a Turbulent Mixing Layer , REZA H. SHEIKHI, MEHDI SAFARI, FATEMEH HADI, Northeastern University — Entropy transport equation is considered in large eddy simulation (LES) of turbulent flows. The irreversible entropy generation in this equation provides a more general description of subgrid scale (SGS) dissipation due to heat conduction, mass diffusion and viscosity effects. A new methodology is developed, termed the entropy filtered density function (En-FDF), to account for all individual entropy generation effects in turbulent flows. The En-FDF represents the joint probability density function of entropy, frequency, velocity and scalar fields within the SGS. An exact transport equation is developed for the En-FDF, which is modeled by a system of stochastic differential equations, incorporating the second law of thermodynamics. The modeled En-FDF transport equation is solved by a Lagrangian Monte Carlo method. The methodology is employed to simulate a turbulent mixing layer involving transport of passive scalars and entropy. Various modes of entropy generation are obtained from the En-FDF and analyzed. Predictions are assessed against data generated by direct numerical simulation (DNS). The En-FDF predictions are in good agreements with the DNS data.

3:33PM D21.00007 Large-eddy simulations of a fully appended submarine model[1] , ANTONIO POSA, ELIAS BALARAS, The George Washington University — In the present study we report large-eddy simulations (LES) the flow around an idealized submarine geometry (DARPA SUBOFF) at a Reynolds number -based on the model length and free stream velocity- equal to 1.2 million. A finite-difference formulation on a cylindrical coordinate grid of 2.8 billion nodes is utilized, and boundary conditions on the submarine model are imposed using an immersed-boundary technique. The boundary layers are "tripped" near the leading edge to mimic the conditions in experiments reported in the literature. Our computations resolve the detailed dynamics of the turbulent boundary layers on the suboff body as well as their interaction with the large scale vortices generated at the sail and fin junctions. The time-averaged velocity profiles in the intermediate wake reach self-similarity, except for the region affected by the wake of the sail. The comparison with the exponential law from the experimental study in the literature is satisfactory. It is also confirmed that the flow coming from the fins causes a deviation from the self-similar profile, which is more evident than in the experiments. Details on the turbulent boundary layer on the surface of the body will be provided, showing a good qualitative agreement with the results in the literature.

[1]Supported by ONR Grant N000141110455, monitored by Dr. Ki-Han Kim

3:46PM D21.00008 Direct numerical simulations of curvature effects on shear layer transition over airfoils[1] , WEI ZHANG, WAN CHENG, ADNAN QAMAR, WEI GAO, RAVI SAMTANEY, King Abdullah University of Science and Technology — Shear layer transition and subsequent turbulent flow development over the leeward section of airfoils are affected by the surface curvature in terms of its associated effects, such as laminar flow separation, adverse pressure gradient, and the interactions between separated flow and wake vortices, etc. We present direct numerical simulations (DNS) of shear layer transitions over two airfoils, NACA 4412 and NACA 0012-64, at 10 deg. angle of attack, and $Re_c = 10^4$ based on uniform inflow velocity and chord length. The two airfoils chosen are geometrically almost the same with identical maximum thickness along with chordwise position but different cambers and hence different curvature. The curvature effects on the flow are presented by the unsteady evolution patterns of laminar flow separation; shear layer detachment, breakdown to turbulence, turbulent boundary layer reattachment and vortex shedding, and quantitative results on the development of turbulent boundary layer are emphasized. This DNS database is generated with an energy conservative fourth-order incompressible Navier-Stokes code with $O(10^9)$ mesh points.

[1]Supported by a KAUST funded project on large eddy simulation of turbulent flows. The IBM Blue Gene P Shaheen at KAUST was utilized for the simulations.

3:59PM D21.00009 Direct Numerical Simulation of a Transient Cumulus Flow , PRASANTH PRABHAKARAN, SURESH DESHPANDE, RODDAM NARASIMHA, JNCASR — Clouds play a major role in climate change, and the ability to simulate moist convection patterns is crictical for prediction of tropical weather and climate. Recent laboratory experiments (Narasimha et al. (2011) PNAS 108.39 (2011): 16164-16169) have successfully reproduced a variety of naturally occuring cloud types and shapes, and throw light on the mechanisms responsible for entrainment and detrainment in cloud flows. Based on this work it was proposed that a 'transient diabatic plume' subjected to off-source diabatic heating is an appropriate model for cumulus flow. In the present work we report the first direct numerical simulation of such a transient diabatic plume, by solving the 3D Navier-Stokes-Boussinesq equations. Visualisation of the cloud flow is carried out using a coarse grid of around 4 million grid points. The final simulation was performed using 128 million grid points at a Reynolds number of 2000. We present the evolution of different flow variables for the transient flow and compare it with a stationary state non-diabatic plume. In particluar, we present results on the dramatic effect of off-source heat addition on the vortical structures in the flow field and on the entraining velocity field.

4:12PM D21.00010 Reynolds number effects on drag reduction of turbulent boundary layers subject to wall oscillation , MANEESH MISHRA, MARTIN SKOTE, School of Mechanical and Aerospace Engineering, Nanyang Technological University, Singapore — Drag reduction (DR) of external flows were studied using direct numerical simulations of spatially growing turbulent boundary layers with temporal wall oscillations. Three simulations with similar oscillation parameters were performed at different streamwise positions to explore the effects of Reynolds number (Re) on DR. One of the simulation cases replicates an experiment and results are in excellent agreement for both mean quantities and turbulence statistics. Downstream development of skin friction, velocity profiles and turbulence statistics have been studied. Spatial transients for the peak values of turbulence statistics have been found to show a non-monotonic behaviour before reaching a stable steady value. To check the feasibility of DR at high Re, a predictive relation has been modelled based on current and previous experimental and simulation data. In light of these results, feasibility of this technique for real world applications is discussed.

Sunday, November 24, 2013 2:15PM - 4:25PM –

Session D22 Turbulence Modeling III 317 - Reetesh Ranjan, Georgia Institute of Technology

2:15PM D22.00001 On the accuracy of simulations of a 2D boundary layer with RANS models implemented in OpenFoam[1] , BENJAMIN J. GRAVES, SEBASTIAN GOMEZ, SVETLANA V. POROSEVA, University of New Mexico — The OpenFoam software is an attractive Computational Fluid Dynamics solver for evaluating new turbulence models due to the open-source nature, and the suite of existing standard model implementations. Before interpreting results obtained with a new model, a baseline for performance of the OpenFoam solver and existing models is required. In the current study we analyze the RANS models in the OpenFoam incompressible solver for two planar (two-dimensional mean flow) benchmark cases generated by the AIAA Turbulence Model Benchmarking Working Group (TMBWG): a zero-pressure-gradient flat plate and a bump-in-channel. The OpenFoam results are compared against both experimental data and simulation results obtained with the NASA CFD codes CFL3D and FUN3D. Sensitivity of simulation results to the grid resolution and model implementation are analyzed. Testing is conducted using the Spalart-Allmaras one-equation model, Wilcox's two-equation k-omega model, and the Launder-Reece-Rodi Reynolds-stress model. Simulations using both wall functions and wall-resolved (low Reynolds number) formulations are considered.

[1]The material is based upon work supported by NASA under award NNX12AJ61A.

2:28PM D22.00002 A hybrid RANS closure scheme for the near-wall turbulence[1] , FARID KARIMPOUR, SUBHAS K. VENAYAGAMOORTHY, Colorado State University — In this study, we propose a parameterization for the eddy viscosity (ν_t) that can be employed in a wall-resolving standard k-ϵ closure model. To this end, we use the equilibrium assumption between the production rate of the turbulent kinetic energy (P) and ϵ in a wall-bounded turbulent flow. Using this assumption and the linear shear stress distribution, the appropriate velocity scale is $U_S = (\epsilon/S)^{1/2}$ while the corresponding length scale is $L_S = f_\mu \kappa y(1-y/\delta)^{3/4}$, where κ is von Kármán's constant, f_μ is van Driest's damping function, y represents the vertical distance from the wall and δ is one half of the channel depth. Consequently, ν_t results as a product of these two characteristic scales, i.e. $\nu_t = U_S L_S$. *'A priori'* tests are performed to assess the validity of the proposed eddy viscosity and the corresponding characteristic scales using the direct numerical simulation (DNS) data of unstratified channel flow. Furthermore, a one-dimensional standard k-ϵ model was developed and '*a posteriori*' tests were performed. The comparison of both '*a priori*' and '*a posteriori*' tests with DNS data show excellent agreement.

[1]Funded by the National Science Foundation

2:41PM D22.00003 Application of the order-of-magnitude analysis to a fourth-order RANS closure for simulating a 2D boundary layer[1] , SVETLANA V. POROSEVA, University of New Mexico — Simulations of turbulent boundary-layer flows are usually conducted using a set of the simplified Reynolds-Averaged Navier-Stokes (RANS) equations obtained by order-of-magnitude analysis (OMA) of the original RANS equations. The resultant equations for the mean-velocity components are closed using the Boussinesq approximation for the Reynolds stresses. In this study OMA is applied to the fourth-order RANS (FORANS) set of equations. The FORANS equations are chosen as they can be closed on the level of the 5th-order correlations without using unknown model coefficients, i.e. no turbulent diffusion modeling is required. New models for the 2nd-, 3rd- and 4th-order velocity-pressure gradient correlations are derived for the current FORANS equations. This set of FORANS equations and models are analyzed for the case of two-dimensional mean flow. The equations include familiar transport terms for the mean-velocity components along with algebraic expressions for velocity correlations of different orders specific to the FORANS approach. Flat plate DNS data (Spalart, 1988) are used to verify these expressions and the areas of the OMA applicability within the boundary layer.

[1]The material is based upon work supported by NASA under award NNX12AJ61A.

2:54PM D22.00004 An implicit turbulence model for Preconditioned-Roe scheme by using Truncated Navier-Stokes Equations , CHUNGGANG LI, MAKOTO TSUBOKURA, Complex Phenomena Unified Simulation Research Team, RIKEN Advanced Institute for Computational Science, Japan — A new turbulence model named dissipative model for Preconditioned-Roe scheme is proposed. The original Roe scheme employs the Roe upwinding dissipation term to stabilize the simulations. In this study, a free parameter ε is used to adjust the Roe upwinding dissipation term appropriately. Based on the procedure developed previously for the Truncated Navier Stokes (TNS) equations and the energy level of small resolved scales, the extra dissipation provided by the dissipative model for the turbulence is meaningful and of physical fundamental, which is the most different from other implicit turbulence models. With the advantages of easy implementation because no extra terms are needed to the equations and the availability on the curvilinear coordinate, the dissipative model is expected to be a promising tool for practical applications.

3:07PM D22.00005 Optimal Turbulence Closures in Galerkin Models , BARTOSZ PROTAS, McMaster University, BERND NOACK, Institut PPRIME, CNRS — In the present study we propose a variational optimization technique to determine an optimal eddy viscosity for a Galerkin model of a fluid flow. Analogously to LES and RANS, such models require suitable closure strategies to account for the effects of unresolved dynamics and ensure stability of long-time integration. A commonly used ansatz involves a linear dissipation term with the magnitude controlled by an eddy viscosity. While the eddy viscosity is often assumed constant or a linear function of the state, there is in fact a lot of evidence that nonlinear eddy viscosities perform better. We show how an optimal form of a nonlinear eddy viscosity can be determined such that the corresponding trajectories of the Galerkin model best match available data. The eddy viscosity is assumed to depend on the fluctuating kinetic energy only, so that our optimal closure results in an autonomous dynamical system. The eddy viscosity is reconstructed in the continuous setting using a non-parametric structure identification method which does not involve any assumptions other than smoothness. The method is applied to a reduced-order model of a mixing layer and the optimal eddy viscosities found reveal nontrivial insights about the behavior of the model.

3:20PM D22.00006 A three-equation bypass transition model based on the intermittency function[1] , XUAN GE, PAUL DURBIN, Iowa State University — An intermittency model that is formulated in local variables is proposed for representing bypass transition in Reynolds-Averaged Navier-Stokes (RANS) computations. No external data correlation is used to fix transition. Transition is initiated by diffusion and a source term carries it to completion. A sink term is created to predict the laminar region before transition and vanishes in turbulent region. For validation of this model, a group of test cases based on flat plate experiments have been set up for numerical simulations in OpenFOAM. It turns out that the current model is capable to predict boundary layer transition on a flat plate both with and without pressure gradients. Decent agreement with the available experiment data is observed.

[1]Supported by NSF Award Number 1228195

3:33PM D22.00007 Use of DNS Data for the Evaluation of Closure Models for Rotating Turbulent Channel Flow , ALAN HSIEH, SEDAT BIRINGEN, ALEC KUCALA, Aerospace Engineering Sciences, University of Colorado at Boulder — A direct numerical simulation (DNS) of a turbulent channel flow rotating about the spanwise axis was conducted at a Reynolds number (based on the centerline velocity and channel half height) 8000, Prandtl number 0.71, and Rossby number 26. Several Reynolds-Averaged Navier-Stokes (RANS) based turbulence models for rotating flows were analyzed and tested. It was shown that the closure approximations in the pressure-strain correlation term proposed by the Speziale, Sarkar, and Gatski (SSG) RSM model were more accurate than the Girimaji EARSM model. The Reynolds stresses, primarily the shear stresses, produced by the Girimaji model were compared to the DNS data and revealed an evident discontinuity in the modeled Reynolds stress profiles; consequently, a smoothing function was generated and applied as a correction so that there is significantly better agreement between the Reynolds shear stress profiles produced by the DNS data and the modified Girimaji model.

3:46PM D22.00008 Separated shear-layer instability reproduction by a Reynolds stress model of turbulence , SUAD JAKIRLIC, ROBERT MADUTA, Darmstadt University of Technology — A boundary layer separating from a solid wall transforms into a 'separated shear layer' exhibiting a broader frequency range. Such a highly-unsteady shear layer separating the mean stream from the flow reversal is dominated by the organized, large-scale coherent structures, influencing to a large extent the overall flow behavior. Unlike in the case of a flat-plate boundary layer separating at a fixed point characterizing a backward-facing step geometry, which can be reasonably well captured by a statistical model of turbulence, the separation process pertinent to continuous curved surfaces as well as some fence- or rib-shaped configurations is beyond the reach of any RANS (Reynolds-Averaged Navier Stokes) model independent of the modeling level. The latter issue motivated the present work, dealing with an appropriate extension of a near-wall Second-Moment Closure (SMC) model towards an instability-sensitive formulation. The production term in the corresponding scale-supplying equation is selectively enhanced through introduction of the ratio of the first to the second derivative of the velocity field, the latter representing the integral part of the von Karman length scale, enabling appropriate capturing of the fluctuating turbulence and accordingly the reproduction of the separated shear-layer instability. The analysis is performed by simulating the flow separated from a fence, an axisymmetric hill and a cylinder configuration.

3:59PM D22.00009 Experimental verification of turbulence models for pressure diffusion process in plane turbulent jet[1] , OSAMU TERASHIMA, YASUHIKO SAKAI, KOUJI NAGATA, YASUMASA ITO, Nagoya University — We performed simultaneous measurement of the three velocity components and the pressure in a plane turbulent jet, and examined turbulence models related to the pressure diffusion process, such as gradient-diffusion model and the model for the rapid/slow terms of the pressure diffusion term. The results show that the gradient-diffusion model developed in the previous studies are valid only in the region where the turbulent intensity and the turbulent/non-turbulent intermittency are high and the production of the turbulent energy is dominant in comparison with other processes such as the convection and diffusion of the turbulent energy in the turbulent energy budget. In addition, it is found that the pressure diffusion of the turbulent energy cannot be modeled accurately by using only the slow term, and its accuracy is improved by considering both rapid and slow terms in the model. This result indicates that modeling the pressure diffusion process using only the slow term has a certain risk leading to a misunderstanding of the turbulent energy transport process.

[1]Grants-in-Aid for scientific research (No. 23760155) and Takahashi Industrial and Economic Research Foundation

4:12PM D22.00010 An alternative eddy-viscosity representation and its implication to turbulence modeling , SUAD JAKIRLIC, Darmstadt University of Technology, Germany, JOVAN JOVANOVIC, Friedrich-Alexander University of Erlangen-Nuremberg, Germany, BRANISLAV BASARA, Advanced Simulation Technologies, AVL List GmbH Graz, Austria — Large majority of turbulence models in the RANS framework (it holds also in the case of the LES method) is based on the eddy-viscosity rationale. The principle task of modeling the Reynolds stress tensor reduces to modeling the eddy-viscosity, representing, according to Boussinesq (1877), the "coefficient of proportionality" between the Reynolds stress and mean rate of strain tensors. In the present contribution an extended formulation based on the least square approach applied to the Boussinesq's correlation is presented. Furthermore, a Taylor-microscale-based formulation is derived originating from the equilibrium assumption related to the equality between the production and dissipation rates of kinetic energy of turbulence. Finally, an expression is proposed reflecting the Reynolds stress anisotropy influence on the eddy-viscosity damping by approaching the solid wall as well as including an appropriate length-scale switch accounting for the viscosity effects through inclusion of the Kolmogorov scales blended with those of the energy-containing eddies. The latter formulation is successfully applied in the framework of an instability-sensitive Reynolds stress model of turbulence. The afore-mentioned eddy-viscosity definitions are comparatively assessed in a series of wall-bounded flow configurations (including separation) in a Reynolds number range.

Sunday, November 24, 2013 2:15PM - 4:25PM –

Session D23 Turbulence: Theory II - General 318 - Sharath Girimaji, Texas A&M University

2:15PM D23.00001 Effects of anisotropy on the fluctuating dissipation scale , IAN MAY, LAKSHMI PRASAD DASI, Colorado State Univerisity — The invariance of the dissipation scale distribution, $Q(\eta)$, to local measures of anisotropy at high Reynolds number is a necessary condition to support the notion of a universal and isotropic small-scale structure of turbulence. We examine the effects of varying levels of anisotropy on $Q(\eta)$ using a monte-carlo approach to model correlated spatial gaussian velocity ensembles. Anisotropy was modeled as a linear variation in velocity rms in space as is the case locally in strongly anisotropic turbulence. $Q(\eta)$ calculated from isotropic simulations matched recent mathematical distributions from the equations of motion and the multifractal formalism. However $Q(\eta)$ in anisotropic cases, where spatially increasing rms was modeled, systematically deviated from the isotropic expectations. Peak locations of $Q(\eta/\eta_0)$ shift left with increasing anisotropy, however a significant reduction in Reynolds number can induce an overall right shift. These results illustrate contrasting effects between local anisotropy and low Reynolds number with respect to the small-scale structure of the dissipative scales of motion.

2:28PM D23.00002 Local dissipation scales in turbulent shear flows , PETER HAMLINGTON, University of Colorado, Boulder — Recent studies of homogeneous isotropic turbulence and wall-bounded shear flows have indicated that dissipation of kinetic energy occurs over a broad range of scales, including scales significantly larger than the classical mean Kolmogorov scale. It is thus possible to construct a field of local dissipation scales by examining the local Reynolds number at every point in a flow. Distributions of the resulting scales have proven to be similar in the flows examined to date, although substantial variations are observed as the wall is approached in turbulent channel flow. These variations could be due to one or several effects in the near-wall region, including decreased Reynolds number, increased flow two-dimensionality, or increased mean shear. In this talk, the effect of mean shear on local dissipation scales is examined by analyzing direct numerical simulations of homogeneously sheared turbulence. The simulations are performed for a range of shear strengths and Reynolds numbers, and the resulting distributions of dissipation scales are compared. The implications of these results for wall-bounded flows are discussed, and the results are also used to test the validity of assumptions concerning local isotropy and scale separation in turbulent shear flows.

2:41PM D23.00003 Nonlocal pressure and viscous contributions to the velocity gradient statistics based on Gaussian random fields[1] , MICHAEL WILCZEK, CHARLES MENEVEAU, Johns Hopkins University — The velocity gradient tensor characterizes the small scales of fully developed turbulence comprehensively. The challenge in understanding its statistical properties in terms of exact statistical evolution equations lies in specifying the nonlocal pressure and viscous effects. Based on the assumption of incompressible Gaussian velocity fields, these statistically unclosed terms are evaluated analytically, and the dynamics of this Gaussian closure and generalizations thereof are discussed and compared to data from direct numerical simulations. The results help to explain how nonlocal pressure Hessian contributions prevent the restricted Euler singularity, and yield insights into the origin of the velocity gradient skewness related to a breaking of the time-reversal symmetry.

[1]Support from a DFG postdoctoral fellowship (WI 3544/2-1) and the US National Science Foundation (CBET 1033942) is gratefully acknowledged.

2:54PM D23.00004 Exploring the link between intermittency in scalar dissipation (χ) and energy dissipation (ε) rates , SIDDHARTHA VERMA, GUILLAUME BLANQUART, California Institute of Technology — The occurrence of spatial and temporal intermittency in χ, analogous to that seen in ε for the velocity field, poses a formidable challenge in the formulation of subgrid scale models for χ. As the scalar transport equation is known to be linear, intermittency in the scalar field must be inherited largely from non-linearity in the momentum equation. This occurrence may be explained physically as the coincidence of steepest gradients in the scalar field (which correspond to the largest magnitudes of χ) with those in the velocity field (largest magnitudes of ε), caused by strong straining of material particles. To determine the extent of the inheritance, we attempt to establish a qualitative as well as quantitative correlation between intermittency in ε and χ. Any external role of the scalar forcing term in the intermittency of χ is also assessed by using two scalar forcing techniques in homogeneous isotropic turbulence, namely mean scalar gradient forcing and linear scalar forcing. A third, unforced configuration, the turbulent mixing layer is used as well, where scalar fluctuations are sustained naturally by a mean gradient present in the cross-stream direction. Appropriate conclusions are also drawn regarding the relevance of the Schmidt number to the extent of intermittency inheritance, in light of the spectral de-linking that happens at very high Schmidt numbers.

3:07PM D23.00005 More on the asymptotic state of high Reynolds number, smooth-wall turbulent flows , DALE PULLIN, California Institute of Technology, ANTHONY LEONARD, California Institute of Technology — This is an update of a hypothesis (Pullin, Inoue & Saito, *Phys. Fluids*, 2013) concerning the asymptotic state of some canonical, smooth-wall turbulent flows. There it was argued, based on the extrapolation to arbitrarily large Reynolds numbers (Re_τ) of both the log-wake law for the mean velocity profile, and also of Townsend-Perry scaling for stream-wise turbulent velocity fluctuations, that over almost all of the turbulent-flow layer, turbulent velocity fluctuations on outer scales asymptotically decline with increasing Re_τ. Presently this is extended to include vorticity fluctuations using scaling proposed by Panton (*Phys. Fluids*, 2009). This suggests that, at least for turbulent channel flow, the asymptotic state consists of vanishingly-small turbulent velocity fluctuations but unbounded enstrophy ($\overline{\omega^2}$) fluctuations on outer scales, over almost the whole turbulent-flow domain.

3:20PM D23.00006 ABSTRACT WITHDRAWN –

3:33PM D23.00007 Why the "K41"/Batcher hypothesis of "local equilibrium" is wrong , WILLIAM K. GEORGE, Imperial College London and Princeton U. — The foundation of modern turbulence theory since Kolmogorov's pioneering paper in 1941 has been the hypothesis that the small scales of the turbulence were in "local" statistical equilibrium relative to those containing most of the energy. This hypothesis is shown to be fundamentally incorrect and internally inconsistent with deductions based upon it, no matter the Reynolds number. In fact deductions from the local equilibrium hypothesis are valid only in flows in strict statistical equilibrium; i.e., flows that are either already statistically stationary at all scales, or in a convective frame, statistically homogeneous. In other words, only in flows where the "local equilibrium" is in fact exact. Hence experiments in such flows (of which there are many) provide no proof at all, contrary to popular belief that "K41" is "proven." Moreover there are many experiments in non-stationary flows that "disprove" "K41," consistent with the conclusions here. The implications of this for a new theory of turbulence are explored using the material derivative of the dissipation rate.

3:46PM D23.00008 What are the origins of -5/3 spectra and related dissipation scalings? , SYLVAIN LAIZET, J. CHRISTOS VASSILICOS, Imperial College London — In this numerical work we present results concerning the spatial development of energy spectra and their associated integral and Taylor scales in conjunction with the spatial developments of vorticity, strain and production rates of vorticity and strain obtained from Direct Numerical Simulations of spatially developing grid-generated turbulence. We use a fractal square grid and a single mesh grid where the mesh is similar to the largest square on the fractal square grid. We find two adjacent but physically different regions in these flows relatively close to the grid: one where the Q-R diagram has not yet formed its well-known, presumed universal, tear-drop shape but where the energy spectra are not too far from a -5/3 shape over a decade of a frequency range which is set by inlet conditions rather than Kolmogorov scalings: and one where the Q-R diagram immediately adopts the well-known tear-drop shape but where the energy spectra are just about proportional to -5/3 over nearly a decade of frequencies. In the present fractal grid simulation, the first region gives rise, as one moves downstream, to the non-equilibrium behaviour $C\varepsilon \approx 1/\mathrm{Re}\lambda$ (see Valente & Vassilicos, PRL, 2012 and Gomes-Fernandes et al., JFM, 2012) whilst the second region leads to $C\varepsilon \approx$ Const.

3:59PM D23.00009 Axisymmetric turbulent wakes with new non-equilibrium similarity scalings[1] , JOHN CHRISTOS VASSILICOS, JOVAN NEDIC, BHARATHRAM GANAPATHISUBRAMANI, Imperial College London, TMFC, IMPERIAL COLLEGE LONDON TEAM — The recently discovered non-equilibrium turbulence dissipation law (Seoud & Vassilicos PoF 19, 2007, Mazellier & Vassilicos PoF 22, 2010, Valente & Vassilicos JFM 687, 2011, Valente & Vassilicos PRL 108, 2012, Gomes-Fernandes et al. JFM 711, 2012) implies the existence of axisymmetric turbulent wake regions where the mean flow velocity deficit decays as the inverse of the distance from the wake-generating body and the wake width grows as the square root of that distance. This behaviour is different from any documented boundary-free turbulent shear flow to date. Its existence is confirmed in wind tunnel experiments of wakes generated by plates with irregular fractal-like edges placed normal to an incoming free stream.

[1]EPSRC

4:12PM D23.00010 Power Fluctuations and Irreversibility in Turbulence , HAITAO XU, MPI for Dynamics & Self-Organization (MPIDS), Goettingen, Germany, ALAIN PUMIR, ENS-Lyon, Lyon, France, GREGORY FALKOVICH, Weizmann Institute of Science, Rehovot, Israel, EBERHARD BODENSCHATZ, MPI for Dynamics & Self-Organization (MPIDS), Goettingen, Germany, MICHAEL SHATS, HUA XIA, NICOLAS FRANCOIS, Australian National University, Canberra, Australia, GUIDO BOFFETTA, University of Torino, Torino, Italy — We show that for fluid turbulence irreversibility manifests itself in the evolution of the kinetic energy of individual fluid elements. We found in experiment and numerical simulations of two-dimensional and three-dimensional turbulence that fluid elements decelerate faster than accelerate. This asymmetry gives rise to negative third moments of energy changes of a fluid element, which we observed to remain constant for time delays in the range characteristic of turbulent eddies, independently of the flow details including space dimensionality. However, turbulence in two and three dimensions show striking differences in how energy is exchanged between fluid elements: pressure forces redistribute energy from fast to slow elements in two dimensions; conversely, in three dimensions, pressure transfers energy from slow to fast ones.

Sunday, November 24, 2013 2:15PM - 4:25PM –

Session D24 Acoustics II 319 - Woutijn Baars, University of Melbourne

2:15PM D24.00001 Acoustophoretic contactless transport and handling of matter[1] , DANIELE FORESTI, MAJID NABAVI, MIRKO KLINGAUF, ALDO FERRARI, DIMOS POULIKAKOS, Laboratory of Thermodynamics in Emerging Technologies, ETH Zurich — Levitation and controlled motion of matter in air, has a wealth of potential applications ranging from materials processing to biochemistry and pharmaceuticals. We present a novel acoustophoretic concept, for the contactless transport and handling of matter in air. Spatiotemporal modulation of the levitation acoustic field allows continuous planar transport and processing of multiple objects (volume 0.1-10 μl). The independence of the handling principle from special material properties (magnetic, optical or electrical) is illustrated with a wide palette of application experiments, such as contactless droplet coalescence and mixing, solid-liquid encapsulation, absorption, dissolution, and DNA transfection. The dynamics of droplets and particles collision is studied numerically and experimentally. The findings show that the secondary acoustic force gives a significant contribution to the samples impact velocity.

[1]We thank the Swiss National Science Foundation (Grant 144397) for financial support.

2:28PM D24.00002 Aeroacoustic sound radiated from a flow past an oscillating and a fixed cylinder in tandem , YUJI HATTORI, IFS, Tohoku University, RYU KOMATSU, Graduate School of Information Science, Tohoku University — The aeroacoustic sound generated in a two-dimensional flow past two circular cylinders in tandem is studied. The upstream cylinder is forced to oscillate transversely, while the downstream cylinder is fixed. This flow is a simplified model of the sound generation due to the interaction of rotating wings and a strut. The sound is captured by direct numerical simulation of the compressible Navier-Stokes equations using the volume penalization method. The amplitude of the sound increases in general with the amplitude and the frequency of the oscillation of the upstream cylinder. However, large reduction of the sound occurs for particular choices of parameter values as the forces acting on the two cylinders are in anti-phase.

2:41PM D24.00003 Adjoint-field errors in high fidelity compressible turbulence simulations for sound control , RAMANATHAN VISHNAMPET, DANIEL BODONY, JONATHAN FREUND, University of Illinois at Urbana-Champaign — A consistent discrete adjoint for high-fidelity discretization of the three-dimensional Navier-Stokes equations is used to quantify the error in the sensitivity gradient predicted by the continuous adjoint method, and examine the aeroacoustic flow-control problem for free-shear-flow turbulence. A particular quadrature scheme for approximating the cost functional makes our discrete adjoint formulation for a fourth-order Runge-Kutta scheme with high-order finite differences practical and efficient. The continuous adjoint-based sensitivity gradient is shown to to be inconsistent due to discretization truncation errors, grid stretching and filtering near boundaries. These errors cannot be eliminated by increasing the spatial or temporal resolution since chaotic interactions lead them to become $O(1)$ at the time of control actuation. Although this is a known behavior for chaotic systems, its effect on noise control is much harder to anticipate, especially given the different resolution needs of different parts of the turbulence and acoustic spectra. A comparison of energy spectra of the adjoint pressure fields shows significant error in the continuous adjoint at all wavenumbers, even though they are well-resolved. The effect of this error on the noise control mechanism is analyzed.

2:54PM D24.00004 Dynamic Mode Decomposition of a Supersonic Jet Exhausting a Convergent-Divergent Nozzle , BERNHARD SEMLITSCH, MIHAI MIHAESCU, LASZLO FUCHS, KTH Mechanics, KTH MECHANICS TEAM[1] — Non-ideal expanded supersonic jets provoke further noise sources additional to the turbulent mixing noise source, which is present in subsonic jets. In particular, the screech tones are a undesired shock associated noise source in the supersonic jet exhaust, since high amplitude acoustic waves are radiated upstream, where the pressure fluctuations could cause damage to the airplane structure. Satisfying suppression of acoustic noise in a supersonic jets requires further investigation of the acoustic noise generation mechanisms. Dynamic Mode Decomposition (DMD) is a flow decomposition method, which is suited to extract the spectral features of the flow-field. Thus, DMD is conceivably capable to extract the spectral flow features, which can lead to the formation of the screech tones. DMD is performed on Large Eddy Simulation data of a Mach 1.56 over-expanded supersonic jet expanding from a convergent-divergent nozzle.

[1]*Linné FLOW Center*

3:07PM D24.00005 Comparison and Properties of Near-Field and Far-Field Events of High Speed Jet[1] , PINQING KAN, JACQUES LEWALLE, Syracuse University — Two independent algorithms are applied to different signals to extract events that are potentially responsible for jet noise production. The data consist of 10 kHz TRPIV measurement and pressure sampling in both near- and far-field. One method uses near-field diagnostics (representatives of 2D velocity sections, e.g. velocity, vorticity, Q criterion, etc.) and near-field and far-field pressure. Applying cross-correlation and continuous wavelet to pairs of these signals, we look for the more dominant events in the time, frequency and lag domain. These are regarded as the main contributors of communication between the selected signals and are recorded as Near-Field Events. The other method only uses far-field microphones. The short time excerpts are identified as Far-Field Events that are common to three FF signals and responsible for peak energy spectrum. To compare these events, we map out their property distribution, including frequency, magnitude and time of occurrence. The individual events are also compared and a high portion is found to be common to both lists. We regard this as a verification of both algorithms.

[1]This work is supported in part by a Syracuse University Graduate Fellowship, by the Department of Mechanical and Aerospace Engineering at SU, and by Spectral Energies LLC, under an SBIR grant from AFRL.

3:20PM D24.00006 Supersonic Jet Noise Reduction Using Microjets , EPHRAIM GUTMARK, DAN CUPPOLETTI, BHUPATINDRA MALLA, University of Cincinnati — Fluidic injection for jet noise reduction involves injecting secondary jets into a primary jet to alter the noise characteristics of the primary jet. A major challenge has been determining what mechanisms are responsible for noise reduction due to varying injector designs, injection parameters, and primary jets. The current study provides conclusive results on the effect of injector angle and momentum ux ratio on the acoustics and shock structure of a supersonic Md=1.56 jet. It is shown that the turbulent mixing noise scales primarily with the injector momentum flux ratio. Increasing the injector momentum flux ratio increases streamwise vorticity generation and reduces peak turbulence levels. It is found that the shock-related noise components are most affected by the interaction of the shocks from the injectors with the primary shock structure of the jet. Increasing momentum flux ratio causes shock noise reduction until a limit where shock noise increases again. It is shown that the shock noise components and mixing noise components are reduced through fundamentally different mechanisms and maximum overall noise reduction is achieved by balancing the reduction of both components.

3:33PM D24.00007 High fidelity measurements in the far-field of a Mach 3 jet , ROMAIN FIÉVET[1], The University of Texas at Austin, WOUTIJN J. BAARS[2], The University of Melbourne, DAVID SILVA[3], CHARLES E. TINNEY[4], The University of Texas at Austin — Recent studies by Baars & Tinney (2012) [APS DFD12-2012-002085] used 1/4inch pressure-field microphones to produce spatial mappings of the far field spectra, OASPL, skewness and kurtosis of the pressure and pressure derivative, as well as other indicators of local and cumulative nonlinear waveform distortion (quadrature spectral density) of the sound field produced by a laboratory-scale Mach 3 jet flow. It was shown that, despite the presence of crackle, cumulative nonlinear distortions were absent along the peak noise path, where such effects have been shown to reside in full-scale studies. The findings were supported by estimates of the Gol'dberg number using relevant jet operating conditions. The experiment of Baars & Tinney is revisited here using higher fidelity instruments (1/8th inch pressure field microphones resolving up to 140kHz +/-1dB) to identify the effects imposed by the larger microphones used by Baars & Tinney (2012).

[1]PhD Student
[2]Post Doctoral Research Fellow
[3]Undergraduate Student
[4]Assistant Professor

3:46PM D24.00008 Acoustic far-field of shroud-lip-scattered instability modes of supersonic co-flowing jets , ARNAB SAMANTA, Indian Institute of Science, JONATHAN B. FREUND, University of Illinois at Urbana-Champaign — We consider the acoustic radiation of instability modes in dual-stream jets, with the inner nozzle buried within the outer shroud, particularly the upstream scattering into acoustic modes that occurs at the shroud lip. For supersonic core jets, several families of instability waves are possible, beyond the regular Kelvin-Helmholtz (K-H) mode, with very different modal shapes and propagation characteristics, which are candidates for changing the sound character of very high-speed jets. The co-axial shear layers are modeled as vortex sheets, with the Wiener-Hopf method used to compute these modes coupled with an asymptotic solution for the far-field radiation. A broadband mode spectra as well as single propagating modes are considered as incident and scattered waves. The resulting far-field directivity patterns are quantified, to show the efficiency of some of these radiation mechanisms, particularly in the upstream direction, which is not directly affected by the Mach-wave-like sound that is radiated from these modes irrespective of any scattering surface. A full Kutta condition, which provides the usual boundary condition at the shroud lip, is altered to examine how vortex shedding, perhaps controllable at the lip, affects the radiated sound.

3:59PM D24.00009 Jet Crackle: Near-field Nonlinear Acoustic Interactions Due to High-Speed Turbulent Sources , DAVID BUCHTA, JONATHAN FREUND, University of Illinois — It is thought that supersonically advecting eddies in high-speed turbulent jets can radiate Mach-wave-like sound, which corresponds with the perception of a rasping or crackling character in their radiated sound. This perception is thought to be in part due to weak shocks in the sound field, but there is another feature of the sound which suggests that more than simple one-dimensional wave steepening occurs: the pressure signals at a point are skewed, with $S_k \geq 0.4$ also correlating to the perception of crackle. This is peculiar because simple nonlinear steepening will not of itself generate skewed pressure signals. We use direct numerical simulations of temporally developing turbulent free shear flows ($Re = 1530$ up to $Re = 2810$ based on momentum thickness) of crackling and non-crackling temporal planar shear layers to study their near-field acoustics and turbulent sound source. Space–time correlations of the turbulence, statistical measures of near-field wave density and propagation directions, and observations of wave-wave interactions suggest that very near-field nonlinear acoustics are potentially important for the generation of these observed peculiar sound-field characteristics.

4:12PM D24.00010 Time Reversal Acoustic in a flowing medium , TRUNG DUNG LUONG, MANISH ARORA, Division of Physics and Applied Physics, School of Physical and Mathematical Sciences (SPMS), Nanyang Technological University, THOMAS HIES, DHI Water & Environment (S) Pte. Ltd., CLAUS-DIETER OHL, Division of Physics and Applied Physics, School of Physical and Mathematical Sciences (SPMS), Nanyang Technological University, CLAUS-DIETER OHL GROUP TEAM, DHI WATER & ENVIRONMENT (S) PTE. LTD. COLLABORATION — We explore the effect of flow on time reversal acoustics (TRA). Traditionally, TRA has been studied in static conditions, while a motion of the medium is expected to degrade the spatio-temporal focussing of the sound pulse. Here, we study the effect of the flow with a TRA system at 1MHz. A controlled flow is added between the emitter and receiver. Additional, a metallic plate is utilized to increases the numerical aperture of the emitting transducer. The impulse response of the non-flowing system, is recorded and time reversed. Then, the response of the hydrophone is recorded in presence and absence of the flow. It is found that the time reversed signal focuses on at the hydrophone in both the cases. In the absence of flow, the focus signal is observed to be shifted in the time domain. Furthermore, there is a drop in the peak-to-peak value of the focus signal in the presence of flow. For a flow rate of 3 cm/s (Re $\sim$ 1000), a distinct shift in the time domain and a reduction of the peak is obtained. The results will be discussed and compared with numerical simulation of TRA under flow conditions.

Sunday, November 24, 2013 2:15PM - 4:25PM –

Session D25 Flow Control II: Jets 320 - Onkar Sahni, Rensselaer Polytechnic Institute

2:15PM D25.00001 Performance Enhancement of a Vertical Tail Using Synthetic Jet Actuators: Flow Physics , NICHOLAS RATHAY, Rensselaer Polytechnic Institute, EDWARD WHALEN, Boeing, MICHAEL AMITAY, Rensselaer Polytechnic Institute — Using aerodynamic flow control techniques, it is possible to reduce the severity or even eliminate the flow separation that occurs over the rudder of a vertical tail on a commercial airplane when it is deflected to high angles. Decreasing the extent of separated flow would result in a larger side force generated by the tail. This could allow for the size of the tail to be reduced, decreasing the overall weight and drag, and potentially creating considerable savings in fuel costs. In this work, wind tunnel experiments were conducted at Rensselaer Polytechnic Institute on a $1/19^{\text{th}}$ scale model of a vertical tail. It was shown that synthetic jet (zero-net-mass-flux) actuators were capable of decreasing the separated flow over the rudder and increasing the side force. Furthermore, Stereo Particle Image Velocimetry was used to understand the interaction of the synthetic jets with the flow over the rudder. The measurements showed regions of reduced and enhanced velocity (relative to the baseline) in the vicinity of the jet trajectory. These regions were believed to in part be the result of an interaction between the synthetic jets' edge vortices and the crossflow. The data suggested that in this application the synthetic jet flow control system could more effective at augmenting side force if the jet orifices were designed or aligned in such a way to modify the weighted contribution of these vortices.

2:28PM D25.00002 Interactions of a finite span synthetic jet with a cross flow , CHIA MIN LEONG, TYLER VAN BUREN, Rensselaer Polytechnic Institute, EDWARD WHALEN, Boeing, MICHAEL AMITAY, Rensselaer Polytechnic Institute, RENSSELAER POLYTECHNIC INSTITUTE TEAM, BOEING COLLABORATION — A synthetic jet is a zero-net-mass-flux flow control actuator that produces alternating ejection and suction of fluid momentum across an orifice. It has been used in numerous applications as an active flow control device to improve aerodynamic performance. Though their aerodynamic performance effects are well known, this present study seeks to understand the fluid dynamic effects of synthetic jets. Specifically, the work investigates the interactions of a finite span synthetic jet with a zero-pressure-gradient laminar boundary layer. This study was performed in a small-scale subsonic wind tunnel with an adjustable test section upper wall that was used to generate a zero-pressure-gradient boundary layer. Several finite span rectangular orifices were chosen for this study. Time and phase-averaged Stereoscopic Particle Image Velocimetry (SPIV) measurements were acquired at multiple planes upstream and downstream of the synthetic jet orifice to explore the interaction of the synthetic jet with the cross flow. The effects of the orifice aspect ratio (12, 18, and 24) and blowing ratio (0.5, 1, and 1.5) were investigated. The unsteady vortical structures observed in the near field and the steady structures in the far field are discussed.

2:41PM D25.00003 Numerical Investigation of Synthetic-jet based Flow Control on Vertical-axis Wind Turbine Blades , ASHWIN MENON, STEVEN TRAN, ONKAR SAHNI, MANE, RPI — Vertical-axis wind turbines encounter large unsteady aerodynamic loads in a sustained fashion due to the continuously varying angle of attack that is experienced by turbine blades during each revolution. Moreover, the detachment of the leading edge vortex at high angles of attack leads to sudden change in aerodynamic loads that result in structural vibrations and fatigue, and possibly failure. This numerical study focuses on using synthetic-jet based fluidic actuation to reduce the unsteady loading on VAWT blades. In the simulations, the jets are placed at the dominant separation location that is observed in the baseline case. We consider different tip-speed ratios, O(2-5), and we also study the effect of blowing ratio (to be in O(0.5-1.5)) and reduced frequency, i.e., ratio of jet frequency to flow frequency (to be in O(5-15)). For all cases, unsteady Reynolds-averaged Navier-Stokes simulations are carried out by using the Spallart-Allamaras turbulence model, where stabilized finite element method is employed for spatial discretization along with an implicit time-integration scheme.

2:54PM D25.00004 Active Flow Control Integrated Diffuser for increased Energy Efficiency in Variable Air Volume Systems , HERMANUS VAN DER SCHIJFF, DAVID MENICOVICH, JASON VOLLEN, MICHAEL AMITAY, Rensselaer Polytechnic Institute — An experimental investigation was performed to study the application of flow control on an HVAC diffuser using synthetic jets distributed evenly along the diffuser edges. The study was conducted on 1:3 scale typical office space (150 ft^2), which included a simulated scale HVAC system supplied by compressed air. Two different jet momentum coefficients were investigated for two inlet flow rates of 40 and 60 CFM. The flow field was measured using hot wire anemometry and Particle Image Velocimetry. Current Variable Air Volume HVAC systems vary the incoming airflow to adjust to changing temperature conditions in the conditioned space. However, when the air flow rate drops below ideal, air distribution becomes inefficient. This study demonstrates the effectiveness of synthetic jets at controlling the incoming airflow and the distribution in the room, showing ability to affect throw coefficient parameters for different flow rates within the test chamber. The use of such devices has the potential to improve air quality and air distribution in building while simultaneously lowering energy demands of HVAC systems.

3:07PM D25.00005 Open loop control of an axisymmetric turbulent wake using pulsed jet blowing[1] , JONATHAN MORRISON, ANTHONY OXLADE, Department of Aeronautics, Imperial College — We investigate the effects of pulsed jet blowing on the turbulent wake of an axisymmetric bullet-shaped body with a sharp trailing edge. The jet is formed from an annular orifice situated immediately below the trailing edge and oriented in the direction of the freestream. By varying the frequency and amplitude of the perturbation, we achieve a mean pressure increase on the base of the body of up to 33%. Modal decomposition of the base-pressure fluctuations reveals a nonlinear coupling between the symmetric ($m = 0$) perturbation and higher order azimuthal modes ($m \pm 1, \pm 2$) that results in an asymmetric mean pressure distribution. The pressure recovery is shown to be proportional to the strength of the jet vortices and is accompanied by a broadband suppression of energy across all modes with no preferential selection, reaching saturation at approximately 5 times the shear layer frequency. This proportionality is a direct result of reduced coupling between the jet perturbation and both the convective and global wake instabilities. The entrainment interface is examined in detail.

[1]We acknowledge financial support of EPSRC grant EP/I005684.

3:20PM D25.00006 Flow Interactions of a Finite-span Synthetic Jet near a Wing Tip , JOSEPH VASILE, MICHAEL AMITAY, Rensselaer Polytechnic Institute — An experimental investigation was performed to study the three-dimensional flow structures and interactions of a finite-span synthetic jet located near the tip of a sweptback finite wing (NACA 4421, AR = 4, $\Lambda = 30^\circ$) at Re = 10^5 and at three angles of attack, 0°, 9° and 15.5°. Three blowing ratios were investigated; Cb = 0.8, 1.2 and 2. Stereoscopic Particle Image Velocimetry data were collected at multiple 2-D planes in the vicinity of the jet's orifice. The effect of the jet's blowing ratio was analyzed using time-averaged and phase-averaged statistics. The study showed that the flow field in the vicinity of the synthetic-jet orifice becomes highly three-dimensional and is governed by the streamwise structures that are associated with the finite span of the jet (edge vortices). Due to the close proximity of the jet to the wing tip, the baseline (i.e., unactuated) flow field is highly three-dimensional with a non-uniform spanwise boundary layer that becomes more pronounced with increasing angle of attack. Consequently, the formation and advection of the secondary flow structures are altered.

3:33PM D25.00007 Feedback Control of the Wake of a Three-Dimensional Blunt Bluff Body , THIBAULT FLINOIS[1], AIMEE MORGANS[2], Imperial College London — When cars or trucks drive on motorways, more than two thirds of their fuel consumption is due to aerodynamic drag, a significant part of which is caused by the large scale separation that takes place near their trailing edge. We tackle this problem using Large Eddy Simulations and use feedback control of synthetic jets to reduce the losses associated with large-scale structures in the wake. The geometry is a long surface mounted block, whose leading edge is not modelled for computational efficiency and the structure of the unforced flow field around this body is similar to the flow over a surface mounted block or hump. Considering this flow field as a control system, the base pressure force was used as the system output and the input is a slot jet actuator located near the trailing edge. Using open-loop forcing, a form drag reduction of about 7.5% was obtained. Open-loop system identification also allowed a transfer function that models the system's response to actuation to be found. Finally, a set of feedback controllers were applied to the plant and their performance was analysed. These controllers successfully reduce the fluctuations in the near wake, with only a small control effort. However, more significant mean drag reductions are expected at higher Reynolds numbers.

[1]PhD Student, Imperial College London
[2]Senior Lecturer, Imperial College London

3:46PM D25.00008 Aerodynamic Flow Control of a Moving Axisymmetric Platform[1] , THOMAS J. LAMBERT, BOJAN VUKASINOVIC, ARI GLEZER, Georgia Institute of Technology — Active fluidic control of induced aerodynamic forces and moments on a moving axisymmetric platform is investigated in wind tunnel experiments. Actuation is effected by controlled interactions between an azimuthal array of integrated synthetic jets with the cross flow to induce localized flow attachment domains over the aft end of the model and thereby alter the global aerodynamic forces and moments. The axisymmetric platform is wire-mounted on a 6 DOF traverse such that each of the eight mounting wires is connected to a servo motor with an in-line load cell for monitoring the wire tension. The desired platform motion is controlled in closed-loop by a laboratory computer. The effects of continuous and transitory actuation on the induced aerodynamic forces of the moving platform are investigated in detail using high-speed PIV. The time-dependent changes in the forces are explored for model maneuvering and stabilization. It is found that the actuation induces forces and moments that are on the order of the forces and moments of the baseline flow. These measurements agree with preliminary results on the stabilization of a model moving in a single DOF demonstrating the effectiveness of the actuation for trajectory stabilization.

[1]Supported by the ARO.

3:59PM D25.00009 Supersonic Jet Noise Reduction Using Flapping Injection and Pulsed Injection[1]

, HAUKUR HAFSTEINSSON, PhD Student, LARS-ERIK ERIKSSON, Professor, DANIEL CUPPOLETTI, PhD Student, EPHRAIM GUTMARK, Professor, DEPARTMENT OF APPLIED MECHANICS, CHALMERS UNIVERSITY OF TECHNOLOGY TEAM, DEPARTMENT OF AEROSPACE ENGINEERING, UNIVERSITY OF CINCINATTI TEAM, SWEDISH DEFENCE MATERIAL ADMINISTRATION, SWEDEN TEAM — Aircraft are in general noisy and there is a high demand for reducing their noise levels. The jet exhaust is in most cases the main noise source of the aircraft, especially for low bypass ratio jet engines. Fluidic injection affecting the shear layer close to the nozzle exit is a promising noise reduction technique as it can be turned of while not needed and thus the negative effect on the engine performance will be minimized. In the presented work, LES is used to compare steady-state mass flow injection with steady-state mass flow flapping jet injection. The work is a direct continuation of a previous LES study on pulsed injection which showed that the pulsed injection induced pressure pulses in the jet which caused increased tonal noise in the downstream directions. The injection system considered in the presented work consists of eight evenly distributed injectors at the nozzle exit plane with a 90° injection angle relative to the flow direction. Flapping jet injection is believed to minimize the creation of these pressure pulses since it provides steady-state mass flow.

[1]This work is funded by Swedish Defense Material Administration (FMV)

4:12PM D25.00010 Drag reduction in a turbulent channel flow using a passivity-based approach

, PETER HEINS, BRYN JONES, University of Sheffield, ATUL SHARMA, University of Southampton — A new active feedback control strategy for attenuating perturbation energy in a turbulent channel flow is presented. Using a passivity-based approach, a controller synthesis procedure has been devised which is capable of making the linear dynamics of a channel flow as close to passive as is possible given the limitations on sensing and actuation. A controller that is capable of making the linearized flow passive is guaranteed to globally stabilize the true flow. The resulting controller is capable of greatly restricting the amount of turbulent energy that the nonlinearity can feed back into the flow. DNS testing of a controller using wall-sensing of streamwise and spanwise shear stress and actuation via wall transpiration acting upon channel flows with $Re_\tau = 100 - 250$ showed significant reductions in skin-friction drag.

Sunday, November 24, 2013 2:15PM - 4:25PM –

Session D26 Reacting Flows II: DNS/LES/RANS 321 - Richard Miller, Clemson University

2:15PM D26.00001 Subgrid-scale mixing of temperature perturbations from flamelet in turbulent partially premixed flames

, SHUAISHUAI LIU, CHENNING TONG, Clemson University — Recent studies have shown that the subgrid-scale (SGS) mixture fraction and temperature in turbulent partially premixed flames have different structures for different SGS scalar variance. For large SGS variance the molecular transport and chemical reaction are tightly coupled while mixing models are greatly based on non-reactive scalars. To account for this coupling effect we use a method proposed by Bilger and Pope [1, 2] to decompose the temperature (a reactive scalar) into a flamelet part and the perturbations from it. The molecular transport of the former is in close form while the latter in unclosed. The diffusion and dissipation of the temperature perturbations are analyzed using high-resolution line images obtained in turbulent partially premixed (Sandia) flames. The results show that for flame regions that are nearly fully burning, the SGS mixing of the temperature perturbations is similar to that of a non-reactive scalar.

[1] R.W. Bilger. Combust. Sci. Tech., 22 (1980) 251-261.
[2] S.B. Pope. Prog. Energy Combust. Sci., 11 (1985) 119-192.

2:28PM D26.00002 Analysis of Subgrid-Scale Backscatter in Turbulent Reacting Flows

, JEFFREY O'BRIEN, JAVIER URZAY, MATTHIAS IHME, PARVIZ MOIN, AMIRREZA SAGHAFIAN, Center for Turbulence Research, Stanford University — In Large-Eddy Simulations of turbulent flows, subgrid-scale (SGS) backscatter of kinetic energy can cause numerical instabilities and the physical mechanism of backscatter is not well understood. While some effort has been made to analyze the phenomenon in inert flows, the behavior of backscatter in reacting flows has been largely unexamined. In this study, Direct Numerical Simulations of inert and reacting supersonic, temporal, hydrogen-air mixing layers are analyzed to assess the effects of compressibility and combustion on SGS backscatter. As in inert, incompressible flows, it is found that a large fraction of the flow domain experiences backscatter at any given time. However, unlike in earlier incompressible studies, the intensity of the backscatter is considerably weaker than that of the forwardscatter such that net backscatter is not observed when averaging in a homogenous direction. In addition, a relationship between the SGS dissipation and eddy viscosity is derived to quantify effects of compressibility. Six allowed combinations of these effects are identified, and their relative frequency is evaluated for both reacting and non-reacting flows. It is found that backscatter occurs preferentially in regions with positive eddy viscosity and local expansion.

2:41PM D26.00003 A Moments-Based Method for Turbulent Combustion Based on Principal Components: A priori and a posteriori validation[1]

, HESSAM MIRGOLBABAEI, TAREK ECHEKKI, North Carolina State University, MECHANICAL AND AEROSPACE ENGINEERING TEAM — Moment-based methods have been widely used in turbulent combustion modeling. These methods are based on the reconstruction of thermo-chemical scalars' statistics from a set of transported moments. This study is concerned with the development of a systematic strategy to construct representative moments using principal component analysis (PCA) and non-linear variants of this method, mainly Kernel PCA and PCA based on a bottleneck artificial neural network. In order to transport the principal components, transport terms must be evaluated and tabulated for a reference problem that spans the desired composition space of the problem of interest. A formulation for these transport terms is developed and validated. *A priori* and *a posteriori* results are presented to validate the proposed moment-based approach and illustrate the computational saving resulting from its implementation.

[1]The work was supported by the National Science Foundation Computational Mathematics Program under grant DMS- 1217200.

2:54PM D26.00004 Modeling of Unsteady Heat Transfer in Flame-Wall Interaction

, HAO WU, MATTHIAS IHME, Stanford University — An extension of the flamelet/progress variable model is developed to include wall-heat loss effects due to convective heat-transfer. The model introduces a source term in the unsteady flamelet equations, which is modeled based on a modified temperature boundary condition of the counter-flow diffusion flame configuration. The thermochemical composition of the resulting non-adiabatic flamelet structure forms a three-dimensional manifold, which is parameterized in terms of mixture fraction, temperature, and scalar dissipation rate. The performance of the model is evaluated in an a priori study of a H_2/O_2 diffusion flame that is stabilized at an inert isothermal wall. Comparisons with DNS-data show that the developed non-adiabatic flamelet model accurately represents conditional and unconditional results for temperature, chemical composition, and wall heat transfer. Following this a priori investigation, the model is applied in LES of a coaxial H_2/O_2 rocket injector, and simulation results from this a posteriori analysis will be compared with experimental data.

3:07PM D26.00005 Unsteady effects on Polycyclic Aromatic Hydrocarbons in a turbulent jet flame , YUAN XUAN, GUILLAUME BLANQUART, California Institute of Technology — Large Eddy Simulations (LES) have been performed on an ethylene/air piloted turbulent sooting jet flame. The current work focuses on capturing the interaction between turbulent transport and the formation of soot precursors. Particular attention is paid to the formation and transport of Polycyclic Aromatic Hydrocarbons (PAH), for their importance in the nucleation process of soot. Given the large time scale related to PAH formation, these species exhibit substantial unsteady effects when subjected to turbulent perturbations. Therefore, transport equations need to be solved for these species along with the Navier-Stokes equations. The chemical source terms are closed using a recently developed linear relaxation model. All other species are assumed to be in steady state and can be evaluated using chemistry tabulation. The mean and variance of velocity components, temperature, and species mass fractions are compared to experimental measurements. Joint probability density functions of soot volume fraction and temperature are particularly analysed. Additional LES results using full chemistry tabulation for all species including PAH are compared to the previous LES results using the PAH relaxation model, to investigate the importance and influence of turbulence-chemistry interaction.

3:20PM D26.00006 Large Eddy Simulation of ignition in an annular multi-injector combustor , RONAN VICQUELIN, MAXIME PHILIP, MATTHIEU BOILEAU, THOMAS SCHMITT, JEAN-FRANÇOIS BOURGOIN, DANIEL DUROX, SÉBASTIEN CANDEL, Ecole Centrale Paris, CNRS EM2C — The present work deals with validating the LES methodology for transient ignition simulations, and in particular elucidating the mechanisms that control the light round sequence in a laboratory annular combustor, representative of many practical industrial systems. The simulation benefits from the unique experimental database built at EM2C on a fully transparent annular chamber equipped with 16 premixed swirled injectors. The F-TACLES combustion model is used for its ability to properly represent the flame propagation.

3:33PM D26.00007 Large-eddy simulations of real-fluid effects in rocket engine combustors , PETER C. MA, JEAN-PIERRE HICKEY, MATTHIAS IHME, Stanford University — This study is concerned with the LES-modeling of real-fluid effects in rocket combustors. The non-ideal fluid behavior is modeled using the Peng-Robinson equation of state, and high-pressure effects on the thermo-viscous transport properties are also considered. An efficient and robust algorithm is developed to evaluate the thermodynamic state-vector. The highly non-linear coupling of the primitive thermodynamic variables in regions near the critical point requires special consideration to avoid spurious numerical oscillations. To avoid these non-physical oscillations, a second-order essentially non-oscillatory (ENO) scheme is applied in regions that are identified by a density-based sensor. The resulting algorithm is applied in LES to a coaxial rocket-injector, and super- and transcritical operating conditions are considered. Simulation results and comparisons with experimental data will be presented, and the influence of boundary conditions on the mixing characteristics will be discussed.

3:46PM D26.00008 Direct Numerical Simulation Study of Nonequilibrium Effects on Mixing and Combustion in Supersonic Jets , HEESEOK KOO, Flight Vehicle Research Center, Seoul National University, VENKAT RAMAN, PHILIP L. VARGHESE, Dept. of Aerospace Engineering and Engineering Mechanics, University of Texas at Austin — Thermochemical nonequilibrium could be significant in scramjet engines due to intense shock-based compression in the pre-combustion isolator region. In particular, vibrational nonequilibrium could adversely affect ignition time and mixing efficiency. To understand the role of nonequilibrium in such flows, direct numerical simulation (DNS) of supersonic flows with vibrational excitation are studied. A linear time-scale model is used to describe the vibrational relaxation of excited species. Essentially, nonequilibrium alters the flow by changing the physical properties that are related to the translational temperature. Such changes introduce nonlinear effect on the scalar mixing process. Further, the redistribution of energy amongst the internal states affects chemical rates. An analysis of the impact of nonequilibrium on combustion is provided.

3:59PM D26.00009 Quasi-dual-mode behavior in the combustor of the HyShot scramjet , JOHAN LARSSON, University of Maryland, RONAN VICQUELIN, Ecole Centrale Paris, JULIEN BODART, Universite de Toulouse, ISAE, IVAN BERMEJO-MORENO, Center for Turbulence Research, STUART LAURENCE, University of Maryland — The flow in the HyShot scramjet combustor is studied using wall-modeled LES with a flamelet combustion model. The focus is on the qualitative changes in the flow structure as the fuel/air equivalence ratio (ER) is increased. For low ERs, an essentially linear pressure-rise is found, consistent with fully supersonic combustion. For higher ERs, a qualitatively different yet stable flow develops, with a stronger shock-train towards the end of the combustor. This shock-train is analogous to what occurs in the isolator in dual-mode operation, but is lodged within the region of heat release, with a position that depends on the ER; this is consistent with recent experimental data obtained by Laurence et al at the German Aerospace Center (DLR).

4:12PM D26.00010 A Priori Analysis of Flamelet-based Modeling for a Dual-Mode Scramjet Combustor , JESSE QUINLAN, JAMES MCDANIEL, University of Virginia, TOMASZ DROZDA, NASA Langley Research Center, GUILHEM LACAZE, JOSEPH OEFELEIN, Sandia National Laboratories — A priori analysis of a dual-mode scramjet combustor is performed using a Reynolds-Averaged Navier-Stokes simulation dataset utilizing finite-rate kinetics to investigate the applicability of flamelet-based combustion models. The HIFiRE Direct Connect Rig flow path is simulated in a dual-mode configuration for the combustion of a JP-7 fuel surrogate using an 18-step chemical mechanism. Simulation results are validated using experimental time-averaged wall pressure measurements. Analysis of the flame structure and combustion mode suggests that combustion regions are predominately of a non-premixed nature and of a high Damkohler number, thereby suggesting the applicability of non-premixed flamelet-based modeling techniques. Regions of premixed combustion were detected but with significantly lower heat-release contribution when compared to the non-premixed combustion regions. Representative flamelet boundary conditions are estimated using an analysis of probability density functions for temperature and pressure. The effects of compressibility on the flame structure and corresponding flamelets are investigated. Insights and future work regarding development of non-premixed flamelet-based models for high-speed compressible flows are discussed.

Sunday, November 24, 2013 2:15PM - 3:33PM –

Session D27 Transonic and Turbomachinery CFD Spirit of Pittsburgh Ballroom A - Scott Morris, University of Notre Dame

2:15PM D27.00001 Transonic Flows of Bethe-Zel'dovich-Thompson Fluids , MARK CRAMER, ALEKSANDR ANDREYEV, Virginia Tech — We examine steady transonic flows of Bethe-Zel'dovich-Thompson (BZT) fluids over thin turbine blades or airfoils. BZT fluids are ordinary fluids having a region of negative fundamental derivative over a finite range of pressures and temperatures in the single phase regime. We present the transonic small disturbance equation, shock jump conditions, and shock existence conditions capable of capturing the qualitative behavior of BZT fluids. The flux function is seen to be quartic in the pressure or density perturbation rather than the quadratic (convex) flux function of the perfect gas theory. We show how this nonconvex flux function can be used to predict and explain the complex flows possible. Numerical solutions using a successive line relaxation (SLR) scheme are presented. New results of interest include shock-splitting, collisions between expansion and compression shocks, two compressive bow shocks in supersonic flows, and the observation of as many as three normal stern shocks following an oblique trailing edge shock.

2:28PM D27.00002 Supersonic flows of a BZT fluid over thin airfoils[1] , FATEMEH BAHMANI, MARK CRAMER, Virginia Tech — We solve a quartic Burgers equation to describe the steady, two-dimensional, inviscid supersonic flow field of a Bethe-Zel'dovich-Thompson (BZT) fluid generated by thin airfoils or turbine blades. A parabolic arc airfoil has been considered. A motivation for this problem is to illustrate the complex flow patterns possible for simple airfoil shapes. The freestream state will be chosen so that the fundamental derivative of gas dynamics is negative for part or even all of the flow. The Burgers equation is solved using the WENO technique. This is the second motivation for this work to demonstrate that the WENO technique is well-suited to the study of BZT fluids. Phenomena of interest include the partial and complete disintegration of compression shocks, the formation of expansion shocks, and the collision of expansion and compression shocks.

[1]This work received support from National Science Foundation Grant CBET-0625015.

2:41PM D27.00003 Entropy Generation in Three-dimensional, Swirling Flows , SCOTT MORRIS, ETHAN PEREZ, JOSHUA CAMERON, ALEKSANDAR JEMCOV, University of Notre Dame — Entropy generation in a transonic, highly loaded, axial turbine stage has been investigated via application of the entropy transport equation. The use of a RANS simulation required additional consideration of entropy transport associated with the time-averaging of products of fluctuating quantities. Analysis of the RANS entropy transport equation allowed the investigators to calculate the volumetric distribution of $\overline{Ds}/\overline{Dt}$; that is, the time rate of change of mean entropy of a material particle. The quantity $\overline{Ds}/\overline{Dt}$ revealed localized regions of entropy increases and decreases of material particles. It was discovered that regions of large $+\overline{Ds}/\overline{Dt}$ occur primarily on the blade suction surface. Additionally, it was found that the passage vortex core, typically associated with high entropy fluid, corresponded with regions of $\overline{Ds}/\overline{Dt} \approx 0$. This result suggests that entropy is generated on the blade suction surface and then collected and convected by the blade passage vortex.

2:54PM D27.00004 The Impact of Casing Geometry on a High Speed Compressor Rotor Passage Shock , MARK ROSS, University of Notre Dame, HAIXIN CHEN, Tsinghua University, JOSHUA CAMERON, SCOTT MORRIS, University of Notre Dame — The adiabatic efficiency of a high speed compressor is inextricably linked with the strength and structure of the rotor passage shock. It is well known that the geometry of the compressor casing can affect the topology of the rotor passage flow field as well as the strength and location of the rotor passage shock. However, partially due to the complexities of this unsteady, swirling, and compressible flow, the community lacks a physics-based understanding of this phenomenon. As a first step in gaining insight into this problem, single-passage RANS simulations of Notre Dame's Stage 04 rotor with four different casing geometries were conducted. The presentation will examine changes in the topology of the rotor passage flow occurring with changes in the casing boundary condition.

3:07PM D27.00005 Global stability analysis of a transonic flow over OAT15A airfoil , FULVIO SARTOR, CLEMENT METTOT, DENIS SIPP, ONERA, DAFE TEAM — A transonic interaction between a shock wave and a turbulent boundary layer on a supercritical profile is numerically and theoretically investigated. If the angle of attack is small, RANS simulations converge towards a steady solution; beyond a critical value, the shock exhibits self-sustained oscillations, and the flow can be related to the so-called transonic buffet. Linear stability analysis indicates that for low angle of attack the flow is stable in a global framework. In this case, the noise amplifier behavior of the flow is investigated through a singular value decomposition of the global Resolvent, which highlights the frequency selection process typical of shock-wave/boundary-layer interactions. It will be shown that the shock behaves as a low-pass filter, and Kelvin-Helmholtz type instability are related to high-frequency unsteadiness. When increasing the angle of attack, an unstable eigenvalue appears and the unsteady behavior can be correctly represented by the unstable global mode, as shown by Crouch et al. JFM 2009. The mechanism that is responsible for buffet onset will be discussed, and comparisons between adjoint/direct global modes and optimal forcing/response will be performed.

3:20PM D27.00006 Mach number effects on compressible flow past a circular cylinder at high Reynolds number , ZHENHUA XIA, YIPENG SHI, ZUOLI XIAO, SHIYI CHEN, State Key Laboratory for Turbulence and Complex Systems, College of Engineering, Peking University, Beijing, 100871 — Compressible flows past a circular cylinder are numerically investigated by using constrained large-eddy simulation (CLES) method at a Reynolds number of 10^6 and with Mach number varying from 0.5 to 0.95. Several mean and statistical quantities, including the drag coefficient, the separation angle, the skin friction coefficient, and the pressure fluctuations are calculated and analyzed. It is found that the separation location moves toward first and then away from the front stagnation point with the increasing Mach number. Further analysis reveals that the fluid flow in the boundary layer on the cylinder changes from turbulent to laminar when the Mach number exceeds a critical value of Mcr $\sim$ 0.65. In other words, laminar boundary-layer separation over the circular cylinder will be observed when the inlet Mach number lies in certain range under consideration. Such a phenomenon is believed to be due to the compressibility effects as Mach number increases.

Sunday, November 24, 2013 2:15PM - 4:25PM –

Session D28 Free-Surface Flows II Spirit of Pittsburgh Ballroom B/C - Enrique Rame, National Center for Space Exploration

2:15PM D28.00001 Free surface shapes in rigid body rotation , ENRIQUE RAME, R. BALASUBRAMANIAM, National Center for Space Exploration Research — When a given volume of fluid in a container spins as a rigid body, the shape of the free surface can be found from the normal stress balance because the pressure field is known up to an additive constant. The properties of periodic (tubular) free surface shapes in rigid rotation have been studied in the past. In this talk we present results for the more practical case of shapes produced by a fixed fluid volume in a cylindrical container of given radius. We will discuss the shapes of vortices spanning the whole cylinder cross section and developing in an infinitely long container, as a function of rotation number. A critical state develops as the rotation rate approaches a critical value, where the vortex depth exhibits a logarithmic asymptotic growth rate with rotation. We will discuss the relation of these critical values to properties previously discovered by D.D. Joseph & colleagues. We will also present properties of shapes in finite-height containers. First, we discuss the case of a vortex of given volume whose free surface contacts the container bottom and wall; next, the case when the free surface contacts the top and bottom lids of a rotating container, in the asymptotic limit of large rotation numbers.

2:28PM D28.00002 The Interaction of a Turbulent Ship-Hull Boundary Layer and a Free Surface[1] , N. MASNADI, N. WASHUTA, A. WANG, J.H. DUNCAN, University of Maryland — The free-surface deformation pattern caused by subsurface turbulent velocity fluctuations in the boundary layer at the mid-length of a naval ship is studied with a novel laboratory scale experimental technique. In this technique, the boundary layer is created in a large tank (13.4 m long, 1.3 m tall, and 2.4 m wide) with a surface-piercing meter-wide stainless steel belt that travels in a horizontal loop around two vertically oriented rollers whose axes are separated by 7.5 m. The device is enclosed in a dry box except for one of the two lengths between the rollers where a straight 6-meter-long section is exposed to the water and represents one side of the ship hull. The belt operates at full-scale ship speeds (up to 15 m/s) in order to match the Reynolds, Froude, and Weber numbers to those of naval ships, thus faithfully modeling the interaction of the turbulence with the free surface at laboratory scale. The water surface profile history midway between the rollers is recorded cinematically in a vertical plane normal to the belt using a Laser Induced Fluorescence (LIF) technique. This surface profile data is used to study the near-wall and far-field frequency content and propagation behavior of the surface ripples.

[1]The support of the Office of Naval Research is gratefully acknowledged.

2:41PM D28.00003 The effect of bed roughness on the free surface of an open channel flow and implications for remotely monitoring river discharge , ERIKA JOHNSON, EDWIN COWEN, Cornell University — The effect of increased bed roughness on the free surface turbulence signature of an open channel flow is investigated with the goal of incorporating the findings into a methodology to remotely monitor volumetric flow rates. Half of a wide (B=2 m) open channel bed is covered with a 3 cm thick layer of loose gravel (D_{50} =0.6 cm). Surface PIV (particle image velocimetry) experiments are conducted for a range of flow depths (B/H=10-30) and Reynolds numbers (Re_H =10,000-60,000). It is well established that bed roughness in wall-bounded flows enhances the vertical velocity fluctuations (e.g. Krogstad et al. 1992). When the vertical velocity fluctuations approach the free surface they are redistributed (e.g. Cowen et al. 1995) to the surface parallel component directions. It is anticipated and confirmed that the interaction of these two phenomena result in enhanced turbulence at the free surface. The effect of the rough bed on the integral length scales and the second order velocity structure functions calculated at the free surface are investigated. These findings have important implications for developing new technologies in stream gaging.

2:54PM D28.00004 Rivulet between two planes: effect of inlet angle[1] , PETER VOROBIEFF, NIMA FATHI, The University of New Mexico — The behavior of gravity-driven rivulets flowing down between two vertical planes has attracted considerable recent attention, driven both by practical interest and by the attractiveness of the problem from the point of view of nonlinear physics. In this investigation, we study the effects of Reynolds number and variations of the inlet boundary conditions on the rivulet flow. The latter include variation in the entrance angle of the inlet with respect to vertical in the plane containing the rivulet. The experimental arrangement allows to create or eliminate fluctuations in the discharge that drives the rivulet, which leads to changes in the flow patterns we observe, including transitions between different flow regimes, and in some cases coexistence of straight and meandering flow. For a wide range of flow regimes, elimination of fluctuations in the discharge rate leads to emergence of stable, straight, non-meandering flow. While a similar observation had been previously made for flows down an inclined plane, this result is interesting, because of differences in the boundary conditions.

[1]This research is partly supported by a gift from the Procter & Gamble Company.

3:07PM D28.00005 Flow around the corner in the water impact problem , ROUSLAN KRECHETNIKOV, University of California at Santa Barbara — In this work we discuss an effect of three dimensionality of the impacting body on the ejecta evolution in the water impact problem as inspired by recent experimental data. The main finding of this study is the revealed and quantified influence of the geometry of a flat plate corner on the ejecta properties. The systematic approach taken here provides a generalization of the classical two-dimensional results.

3:20PM D28.00006 Oblique impact of dense granular sheets , JAKE ELLOWITZ, NICHOLAS GUTTENBERG[1], HEINRICH M. JAEGER, SIDNEY R. NAGEL, WENDY W. ZHANG, Department of Physics and the James Franck Institute, University of Chicago, Chicago, IL 60637 — Motivated by experiments showing impacts of granular jets with non-circular cross sections produce thin ejecta sheets with anisotropic shapes, we study what happens when two sheets containing densely packed, rigid grains traveling at the same speed collide asymmetrically. Discrete particle simulations and a continuum frictional fluid model yield the same steady-state solution of two exit streams emerging from incident streams. When the incident angle $\Delta\theta$ is less than $\Delta\theta_c = 120^\circ \pm 10^\circ$, the exit streams' angles differ from that measured in water sheet experiments. Below $\Delta\theta_c$, the exit angles from granular and water sheet impacts agree. This correspondence is surprising because 2D Euler jet impact, the idealization relevant for both situations, is ill posed: a generic $\Delta\theta$ value permits a continuous family of solutions. Our finding that granular and water sheet impacts evolve into the same member of the solution family suggests previous proposals that perturbations such as viscous drag, surface tension or air entrapment select the actual outcome are not correct.

[1]Currently at Department of Physics, University of Oregon, Eugene, OR 97403

3:33PM D28.00007 Simultaneous Multiphase PIV of Capillary Waves on a High Velocity Liquid Jet , MATTHIEU ANDRE, PHILIPPE BARDET, The George Washington University — Relaxation of a laminar boundary layer below the free surface of a jet is inviscidly unstable and can roll-up which generates millimeter size waves. The latter largely modify important characteristics of jets such as heat and mass transfers between phases and can lead to breakup, or air entrainment. Two dimensional linear stability analysis predicts the initial disturbance wavelength and growth rate for inviscid flows; it does not take into account the effects of viscosity, non-linearity, or actual boundary layer profile. Because of the small temporal and spatial scales associated with this flow, few experimental data are available. Data acquisition is further complicated by the presence of a free surface with steep waves. The current experiment consists in a 20.3 mm $\times$ 146.0 mm water slab laminar jet flowing onto a transparent open-channel at a Reynolds number of 2.9 $\times$ 104 to 1.4 $\times$ 105. Two high speed cameras are employed to obtain velocity fields simultaneously in the liquid and in the gas phase with Particle Image Velocimetry (PIV). Fluorescent dye is added in the liquid in order to improve interface detection. Each phase is recorded at 10 kHz, leading to a temporal resolution of 100 μs and high magnification lenses give a spatial resolution of 200 μm. The results confirm the mechanism of formation of the short surface waves. Generation of surface vorticity is identified in high curvature regions. Knowledge of the velocities in both phases allows studying vorticity flux through the free surface. The latter stage of wave growth can be accompanied by the formation of a vortex pair in the liquid and air entrapment.

3:46PM D28.00008 Ejecta evolution during cone impact , JEREMY MARSTON, IVAN VAKARELSKI, SIGURDUR THORODDSEN, King Abdullah University of Science and Technology — We present results from an experimental study of the impact of conical shaped bodies into a pool of liquid. By varying the cone angle, impact speed and liquid physical properties, we examine a broad parameter space and seek to find conditions when self-similarity can be observed during this phenomena. We use high-speed imaging to capture the early-time motion of the liquid ejecta which emanates from the tip of the cone and travels up along the cone surface. Surprisingly, we find that the detachment of the ejecta can be simply described by air entrainment relationships derived from coating experiments.

3:59PM D28.00009 Flow-pattern analysis in open and closed square ducts: A comparative investigation of corner vortices[1] , JOCHEN KRIEGSEIS, Intitute of Fluid Mechanics, Karlsruhe Institute of Technology, MARKUS VAAS, Institute for Hydromechanics, Karlsruhe Institute of Technology, BETTINA FROHNAPFEL, Intitute of Fluid Mechanics, Karlsruhe Institute of Technology — In the present study secondary flows in straight square ducts are investigated experimentally for open and closed geometries. The flow of the closed square duct typically consists of a set of eight equal-sized counter-rotating vortices. In contrast, the flow in an open flume of (identical) square geometry is considered, where the development of secondary vortices is strongly affected by the presence of the free surface. Stereo Particle Image Velocimetry (SPIV) experiments have been performed so as to measure the flow in both geometries for varying Reynolds numbers. From the resulting 2D3C velocity information secondary vortices (superimposed onto the mean primary flow) are identified. As expected, the upper bisectors' flow topology differs significantly between open and closed ducts, where typical inner and outer vortices are identified from the open-duct data. Interestingly, the secondary vortices of the lower bisectors of both duct flows reveal a similar topology. In order to study this seeming similarit more rigorously, a modal analysis of the respective flow data is performed by means of proper orthogonal decomposition (POD). As such, similarities and limitations of the comparability of the respective corner vortices are uncovered and discussed as function of Reynolds number.

[1]J. Kriegseis acknowledges financial support by the German Academic Exchange Service (DAAD)

4:12PM D28.00010 Water entry without surface seal: extended cavity formation , MOHAMMAD MANSOOR, JEREMY MARSTON, IVAN VAKARELSKI, SIGURDUR THORODDSEN, King Abdullah University of Science and Technology — We report results from an experimental study of cavity formation during the impact of hydrophobic spheres onto water. Using a splash-guard mechanism, we block the spray during initial contact from closing thus eliminating the phenomenon known as "surface seal," which typically occurs at $Fr = O(100)$. As such, we are able to observe the evolution of a smooth cavity in an extended parameter space than has been achieved in previous studies. By systematically varying the tank size and sphere diameter, we examine the influence of increasing wall effects on these guarded impact cavities and note the formation of surface waves with wavelength $\lambda = O(10)$ cm and acoustic waves $\lambda_a = O(D_0)$along the cavity interface, which produce multiple pinch-off points. Acoustic waves are initiated by pressure perturbations, which themselves are generated by the primary cavity pinch-off. Using high-speed particle image velocimetry (PIV) techniques we show the larger waves ($\lambda = O(10)$ cm) have a standing nature. We show that previously deduced scalings for the normalised (primary) pinch-off location $H_p/H = 1/2$ and time $\tau \propto (R_0/g)^{1/2}$ do not hold in the presence of strong wall effects for $D_0/D_{tank} \geq 1/16$.

Sunday, November 24, 2013 2:15PM - 3:20PM –

Session D30 Porous Media Flows II: Mixing and Turbulence 408 - James Liburdy, Oregon State University

2:15PM D30.00001 Stretching, Coalescence and Mixing in Porous Media , TANGUY LE BORGNE, University of Rennes 1, CNRS UMR 6118, France, MARCO DENTZ, CSIC, IDAEA, Barcelona, Spain, EMMANUEL VILLERMAUX, Aix Marseille Université — We study scalar mixing in heterogeneous conductivity fields, whose structural disorder varies from weak to strong. A range of stretching regimes is observed depending on the level of structural heterogeneity, measured by the log-conductivity field variance. We propose a unified framework to quantify the overall concentration distribution predicting its shape and rate of deformation as it progresses towards uniformity in the medium. The scalar mixture is represented by a set of stretched lamellae whose rate of diffusive smoothing is locally enhanced by kinematic stretching. Overlap between the lamellae is enforced by confinement of the scalar line support within the dispersion area. Based on these elementary processes, we derive analytical expressions for the concentration distribution, resulting from the interplay between stretching, diffusion and random overlaps, holding for all field heterogeneities, residence times, and Peclet numbers.

2:28PM D30.00002 Solute Blob Evolution and Mixing Dynamics in a Darcy Scale Heterogeneous Porous Medium , MARCO DENTZ, IDAEA-CSIC, TANGUY LE BORGNE, University of Rennes, FELIPE DE BARROS, USC — We study the mixing behavior of a dissolved substance that evolves from a solute blob in a two-dimensional heterogeneous porous medium. The study scale is mesoscopic so that flow is governed by Darcy's law. Heterogeneity is induced by spatially variable permeability. The fundamental mechanism governing the evolution and mixing dynamics of a solute blob are the competition of the stretching action of a material line and diffusion. We formulate the transport problem in a Lagrangian framework and consider the motion of solute particles that form the blob, in the coordinate system attached to the material element on which it originates. The blob evolution is fully characterized by the stochastic time series of stretching and shear rates of the material segment in its own coordinate system. Theses stochastic series are investigated numerically using random wak particle tracking simulation. In this stochastic framework, we study the ensemble concentration PDF, concentration entropy and scalar dissipation rate. The aim is to relate the mixing properties to the appearance of coherent structures as quantified by the Okubo-Weiss measure and its Lagrangian counterpart.

2:41PM D30.00003 Scale Estimation for Turbulent Flows in Porous Media[1] , VISHAL PATIL, JAMES LIBURDY, Oregon State University — Flow in porous media, once extended into the turbulent flow regime can become very complex due to the nature of the flow geometry and related scales of motion. The ability to model porous media turbulence flow has been hampered by the inability to develop an appropriate understanding of the complexities associated with the impact of pore scale dynamics on the overall turbulence contributions to dispersion and mixing. In this paper we use direct PIV measurements of the turbulence within a randomly packed porous bed of uniform size spheres to better understand scaling distributions. Refractive index matching was used to obtain time resolved velocity vector data within specific pores to compare turbulence quantities versus pore Reynolds numbers. Results are used to determined the characteristics of scales associated with velocity, length and time. The large scale events, within the domain of the pore size are evaluated based on correlation functions within the pore. In addition, estimates of the Komolgorov scales are presented versus pore Re based on integral scale results. The relationships between characteristics pore sizes, pore Re, the integral scales and turbulent statistics are presented and shown to reach an asymptotic limit for large pore Re.

[1]This study was supported in part by NSF through grant 0933857 under the Particulate and Multiphase Processing Program, Dr. Ashok S. Sangani Program Mananger, and is gratefully acknowledged.

2:54PM D30.00004 Direct Numerical Simulation of a turbulent channel flow over Slippery Liquid-Infused Porous Surfaces[1] , ISNARDO ARENAS, University of Puerto Rico at Mayaguez, PAOLO ORLANDI, Universita di Roma La Sapienza, STEFANO LEONARDI, Dept. Mechanical Engineering, University of Texas at Dallas — Direct Numerical Simulations of two superposed fluids in a turbulent channel have been performed at Re ranging from 180 to 400. With respect to previous studies in the present numerical simulation both the flow inside the porous media and the overlying flow has been resolved. Three different substrates have been considered: longitudinal and transverse square cavities and array of circular cylinders. A tracking interface algorithm has been developed using the level set technique. The velocity profiles at the interface present a kink, which is due to the different viscosity. In fact, at the interface the stress is the same in the two fluids and then to a larger viscosity it corresponds a smaller gradient of velocity. Surface tension decreases the turbulence levels consequently, a drag reduction of about 15% can be observed. The stability of the interface is crucial to achieve drag reduction. Even for higher viscosity near the wall, drag reduction is observed. This should be due to the suppression of wall normal velocity fluctuations and to a decrease of turbulent production at the interface. The value of the viscosity inside the patterned surface appears to be less critical than the stability of the interface to achieve drag reduction.

[1]This research was supported by ONR N00014-12-01-0875 andN00014-12-01-0962.

3:07PM D30.00005 Dynamics of temporally-evolving shear layers on the interface between a porous medium and a pure fluid[1] , PANAGIOTIS D. ANTONIADIS, MILTIADIS V. PAPALEXANDRIS, Université catholique de Louvain — In this talk we present results from our study on the dynamics of flows at the macroscopic interface between a porous medium and a pure fluid. To this end, we employ a variation of the unsteady Darcy-Brinkman equation, which is valid both inside and outside the porous medium. The major advantage of this approach is that it does not require additional interface conditions. In the first part of the talk, we present a linear stability analysis for unbounded shear layers on the interfaces of interest. According to our analysis, these layers are unconditionally unstable, regardless of the porosity of the medium. Subsequently, we present results of three-dimensional simulations of such shear layers. According to these simulations, the velocity gradients across the interface result in the onset of a Kelvin-Helmholtz instability which grows over time, leading to spanwise roller formation and pairings. There is also concurrent formation of thin "rib" vortices, as in the case of single-phase plane mixing layers. Important characteristics of the flow, such as self-similarity and growth rate of the shear layer, are also discussed.

[1]This work is supported by the National Fund for Scientific Research (FNRS), Belgium.

Sunday, November 24, 2013 2:15PM - 4:25PM –

Session D31 Porous Media Flows III: Wicking, Drying and Displacement of Immiscible Fluids

402 - Daniel Anderson, George Mason University

2:15PM D31.00001 Asymmetric Wicking and Reduced Evaporation Time of Droplets Penetrating a Thin Double-Layered Porous Material , ARIA VAHDANI, California Institute of Technology, AMIR GAT, Technion - Israel Institute of Technology, ALBERT NOWAKOWSKI, HOMAYUN NAVAZ, Kettering University, MORTEZA GHARIB, California Institute of Technology — We study numerically and experimentally the penetration and evaporation dynamics of droplets wicking into a thin double-layered porous material with order-of-magnitude difference in the physical properties (such as capillary pressure drop, porosity or permeability) between the layers. We show that such double-layered porous materials can be used to create highly asymmetrical wicking properties, preventing liquid droplets wicking from one surface to the other, while allowing for wicking in the reverse direction. In addition, these double-layered porous materials are shown to reduce the evaporation time of droplets penetrating into the porous surface, compared with a single-layered material of equal thickness and physical properties similar to either of the layers. The asymmetric wicking and reduced evaporation time demonstrated in such double-layered porous materials may be of interest to applications such as medical bandages and wearable fabrics.

2:28PM D31.00002 How books are wet by water , JUNGCHUL KIM, HO-YOUNG KIM, Seoul National University — It is well known that a sheet of paper, a hydrophilic porous medium, imbibes water via capillary action. The wicking on two-dimensional sheets has no preferred direction, in general. However, when water is spilled on a book, a number of pieces of paper fastened together on one side, we notice that corners are wet first compared to the rest of the area. This is because the wicking along the sharp corner experiences weaker resistance than that into pores within paper. We study a simple model of this wicking dynamics in the context of the surface-tension-driven vertical rise of a liquid along a corner of folded paper. We find that the liquid height at the corner follows a power law different from that at the corner formed by impermeable walls (A. Ponomarenko, D. Quere, and C. Clanet, J. Fluid Mech. 666, 146-154, 2011). The difference is caused by the fact that the Laplace pressure that drives the vertical rise is independent of the liquid height on permeable walls (paper) while it increases with height at the corner of impermeable walls. The experiments are shown to be consistent with our theory

2:41PM D31.00003 Experiments versus modeling of buoyant drying of porous media , DOMINIQUE SALIN, University Pierre & Marie Curie, ANDREAS YIOTIS, Institut Demokritos (Athens), ESHAN TAJER, YANNIS YORTSOS, University of Southern California — A series of isothermal drying experiments in packed glass beads saturated with hydrocarbons are conducted. The transparent cell allow observation of the formation of liquid films, as the gaseous phase invades the pore space. We demonstrate the existence of an early Constant Rate Period that lasts as long as the films saturate the surface of the packing, and of a subsequent Falling Rate Period that begins practically after the detachment of the film tips from the external surface. During the CRP, the process is controlled by diffusion within the stagnant gaseous phase in the upper part of the cells. During the FRP, the process is controlled by diffusion within the packing, with a drying rate inversely proportional to the observed position of the observed tips in the cell. Our model incorporates effects of corner film flow, internal and external mass transfer and the effect of gravity. Analytical results were derived. We are thus able to obtain results for the drying rates, the critical saturation and the extent of the film region with respect to the various dimensionless numbers that describe the process; the Bond, Capillary numbers and the dimensionless extent of the mass boundary layer. The experimental results agree very well with the theory, provided that the latter is generalized to account for the effects of corner roundness in the film region which were neglected in our analytical approach.

2:54PM D31.00004 Homogenization Approaches for Draining in Layered Porous Media[1] , DANIEL ANDERSON, George Mason University — Motivated by the problem of gravity currents in heterogeneous porous media, we examine the problem of gravitational drainage through a layered porous medium. In particular, for a one-dimensional drainage problem we focus on free boundary motion through layered media. We examine analytical and numerical solutions as well as ones generated by asymptotic approximations schemes that may prove useful in more general settings. Of particular interest is the identification of corrections to the leading-order approximations based on homogenization theory.

[1]Supported by NSF DMS-1107848

3:07PM D31.00005 Influence of heterogeneity on second-kind self-similar solutions for gravity currents , ZHONG ZHENG, IVAN CHRISTOV, HOWARD STONE, Department of Mechanical & Aerospace Engineering, Princeton University, COMPLEX FLUID GROUP TEAM — We report experimental, theoretical and numerical results on the effects of horizontal heterogeneity on the propagation of viscous gravity currents. We use two geometries to highlight these effects: (*a*) a horizontal channel (or crack) whose gap thickness varies as a power-law function of the streamwise coordinate; (*b*) a heterogeneous porous medium whose permeability and porosity have power-law variations. We demonstrate that two types of self-similar behaviors emerge as a result of horizontal heterogeneity: (*a*) a first-kind self-similar solution is found using dimensional analysis (scaling) for viscous gravity currents that propagate away from the origin (point of zero permeability); (*b*) a second-kind self-similar solution is found using a phase-plane analysis for gravity currents that propagate toward the origin. These theoretical predictions, obtained using the ideas of self-similar intermediate asymptotics, are compared to experimental results and numerical solutions of the governing partial differential equations developed under the lubrication approximation. All three results are found to be in good agreement.

3:20PM D31.00006 Pulsed-pressure driven displacement of a non-Newtonian fluid in a radial Hele-Shaw cell , CAROLINE PEREIRA, ANDREW WHITE, THOMAS WARD, Iowa State University — Displacing non-Newtonian fluids in porous media is an extremely challenging problem. While Newtonian fluids typically experience fingering instabilities, non-Newtonian fluids yield dendritic type fingering patterns. In this talk we present experimental data for the displacement of a finite volume of viscoelastic liquid by using pulsed-pressure driven gas flow. Experiments are performed using a radial Hele-Shaw cell at gap spacings ranging from 50-200 microns. The viscoelastic liquids are a mineral oil mixed with high molecular weight poly-isobutylene (M.W. 4.7 million) at concentrations 100-1000 ppm. Air injection pressures range from 0.1-0.5 psi and pulse frequencies range from 0.1-10 Hz. Analysis of the finite liquid volume allows for measurement of the residual film thickness. Also, the gas expansion rate as a function of the pulse frequencies will be presented. The experiments reveal a clear correlation between the pulse frequency, film formation and stability (as measured by the finger formation rate) for a wide range of experimental parameters.

3:33PM D31.00007 Immiscible fluids in mixed wet porous media: the role of wettability correlations , JULIE MURISON, Max Planck Insitute of Dynamics and Self Organisation, BENOIT SEMIN, Laboratorie de Physique Statistique, JEAN-CHRISTOPHE BARET, STEPHAN HERMINGHAUS, MATTHIAS SCHROETER, MARTIN BRINKMANN, Max Planck Insitute of Dynamics and Self Organisation — Various phenomena observed during immiscible displacement in a porous medium can be related to pore wall wettability. Petroleum engineers traditionally quantify the overall wettability of a rock sample in terms of the Ammot-Harvey or USBM index. To establish a link between these gloabl quantities and the pore-scale distribution of surface energies, we developed a series of model porous media. Using a variety of preparation methods, we are able to create dense beds of glass beads with the same average surface energy, differing only in the typical extension of the wetting and non-wetting surface domains. Experimental measurements of capillary pressure saturation curves for repeated imbibition and drainage show that the work dissipated in a complete cycle is monotonically increasing with the correlation length ξ of the surface energies. To test whether capillary hysteresis can be linked to specific features of the front morphology, we visualized the distribution of liquids by means of X-ray microtomography. The Minkowski measures volume, surface area, and Euler number are employed to characterize the interfacial shape. Differences of the front morphology during imbibition and drainage match with trends observed for the hysteresis loop opening.

3:46PM D31.00008 A new computational technique for modeling underground reservoirs , AMIR ARYA, Politecnico Di Torino — The water-oil micro-emulsion in underground reservoirs is considered as one of the most important processes in enhanced oil recovery. The water flow in many natural reservoirs is characterized by density gradients due to the variation of temperature or salinity. Despite this, the effect of stratification on the oil recovery is yet to be explored. Here, we demonstrate the importance of the density stratification on the water-oil interaction in underground reservoirs. We use a system of a drop in a cavity as an idealized model simulating the oil droplets in the rock of the oil. We perform a comprehensive parametric study in order to characterize combined effects of buoyancy, inertia, and density diffusion on the viscous retardation and displacement efficiency of the system. In addition we introduce a new computational technique to efficiently model the process.

3:59PM D31.00009 3d Forced multiphase flow on the pore scale[1] , HAGEN SCHOLL, Experimental Physics, Saarland University, D-66041 Saarbrücken, Germany, KAMALJIT SINGH, Max Planck Institute for Dynamics and Self-Organization, D-37077, Göttingen, Germany, MARIO SCHEEL, MARCO DIMICHIEL, European Synchrotron Radiation Facility, BP 220, F-38043 Grenoble, France, STEPHAN HERMINGHAUS, Max Planck Institute for Dynamics and Self-Organization, D-37077, Göttingen, Germany, RALF SEEMANN, Experimental Physics, Saarland University, D-66041 Saarbrücken, Germany — Using ultra fast x-ray tomography the forced imbibition of an aqueous phase into an initially oil filled matrix is studied. The water is volume controlled flushed into cylindrical columns filled with oil saturated spherical bead packs. The oil displacement is imaged in real time having a spacial resolution of 11 microns and a temporal resolution of about 1 second. To clearly distinguish the aqueous from the oily phase a contrast agent was added to the aqueous phase. The influence of wettability, oil viscosity, gravity and flow velocity was explored and analyzed in terms of temporal development of oil saturation and front shape. It turned out that capillary forces are the key to understand the forced multiphase behavior in the explored parameter range.

[1]Funding was provided by the BP-ExploRe project.

4:12PM D31.00010 Mathematical modeling heat and mass transfer processes in porous media , DARKHAN AKHMED-ZAKI, al-Farabi Kazakh National University — On late development stages of oil-fields appears a complex problem of oil-recovery reduction. One of solution approaches is injecting of surfactant together with water in the form of active impurities into the productive layer – for decreasing oil viscosity and capillary forces between "oil-water" phases system. In fluids flow the surfactant can be in three states: dissolved in water, dissolved in oil and adsorbed on pore channels' walls. The surfactant's invasion into the reservoir is tracked by its diffusion with reservoir liquid and mass-exchange with two phase (liquid and solid) components of porous structure. Additionally, in this case heat exchange between fluids (injected, residual) and framework of porous medium has practical importance for evaluating of temperature influences on enhancing oil recovery. Now, the problem of designing an adequate mathematical model for describing a simultaneous flowing heat and mass transfer processes in anisotropic heterogeneous porous medium –surfactant injection during at various temperature regimes has not been fully researched. In this work is presents a 2D mathematical model of surfactant injections into the oil reservoir. Description of heat- and mass transfer processes in a porous media is done through differential and kinetic equations. For designing a computational algorithm is used modify version of IMPES method. The sequential and parallel computational algorithms are developed using an adaptive curvilinear meshes which into account heterogeneous porous structures. In this case we can evaluate the boundaries of our process flows – fronts ("invasion", "heat" and "mass" transfers), according to the pressure, temperature, and concentration gradient changes.

Sunday, November 24, 2013 2:15PM - 4:25PM –

Session D32 Surface Tension Effects II: Interfacial Flows 403 - Karen Daniels, North Carolina State University

2:15PM D32.00001 The Elasto-capillary Landau-Levich Problem[1] , GEORGE HOMSY, HARISH DIXIT, University of British Columbia — We consider the dip-coating flow problem when the interface has both an elastic bending stiffness and a constant surface tension. In the case where interfacial tension is negligible, we assume the elasticity number El - the ratio of surface elasticity to viscous forces - is small and develop the solution for the free boundary as a matched asymptotic expansion in powers of $El^{1/7}$, thus determining the film thickness as a function of El. A remarkable aspect of the problem is the occurrence of multiple solutions, and five of these are found numerically. In any event, the film thickness varies as $El^{4/7}$, or equivalently, $U^{4/7}$, where U is the plate speed, in agreement with previous experiments. The solution for the elasto-capillary problem is formulated in a similar way, with an elasto-capillary number, ϵ, (the ratio of elasticity to surface tension), as an additional parameter. It is possible to connect the problems of pure elasticity and elasto-capillarity respectively through the parameter ϵ, but connecting one of the five elasto-capillary branches to the classical Landau-Levich result result remains an elusive goal.

[1]We gratefully acknowledge funding from the Natural Science and Engineering Research Council (NSERC) of Canada.

2:28PM D32.00002 Capillarity-Driven Bubble Separations[1] , ANDREW WOLLMAN, MARK WEISLOGEL, Portland State University, MICHAEL DREYER, ZARM — Techniques for phase separation in the absence of gravity continue to be sought after 5 decades of space flight. This work focuses on the fundamental problem of gas bubble separation in bubbly flows through open wedge-shaped channel in a microgravity environment. The bubbles appear to rise in the channel and coalesce with the free surface. Forces acting on the bubble are the combined effects of surface tension, wetting conditions, and geometry; not buoyancy. A single dimensionless group is identified that characterizes the bubble behavior and supportive experiments are conducted in a terrestrial laboratory, in a 2.1 second drop tower, and aboard the International Space Station as part of the Capillary Channel Flow (CCF) experiments. The data is organized into regime maps that provide insight on passive phase separations for applications ranging from liquid management aboard spacecraft to lab-on-chip technologies.

[1]NASA NNX09AP66A, NASA Oregon Space Grant NNX10AK68H, NASA NNX12AO47A, DLR 50WM0535/0845/1145

2:41PM D32.00003 Inertial and Washburn Regimes in Filling of Charged Capillaries , SIDDHARTHA DAS, SUSHANTA K. MITRA, Department of Mechanical Engineering, University of Alberta, Edmonton, Alberta, Canada T6G 2G8, J.C.T. EIJKEL, BIOS, The Lab-on-a-Chip Group, MESA+ Institute for Nanotechnology, University of Twente, P.O. Box 217, 7500 AE Enschede, The Netherlands, N.R. TAS, Transducers Science and Technology, MESA+ Institute for Nanotechnology, University of Twente, P.O. Box 217, 7500 AE Enschede, The Netherlands, SUMAN CHAKRABORTY, Department of Mechanical Engineering, Indian Institute of Technology, Kharagpur-721302 — We discuss the filling dynamics of charged capillaries. Presence of charge on the capillary walls leads to formation of an Electric Double Layer (EDL) at the liquid-capillary-wall interface. Migration of the charge density of the EDL during capillary filling leads to two distinct effects - on one hand it reduces the capillary drive by triggering an opposing electrical force, while on the other hand the induced electroosmotic transport causes a reduction in the net drag force. The ultimate result is a capillary filling process with reduced filling speed, with altered inertial and Washburn regimes, as well as the criteria that dictates the transition between these two regimes.

2:54PM D32.00004 Impact on Floating Membranes , NICOLAS VANDENBERGHE, LAURENT DUCHEMIN, IRPHE, Marseille, France — We report on an experiment focusing on the wave dynamics triggered by the impact of a sphere on a floating elastic membrane. The thin rubber elastic sheet floats on a pool of water. After impact two distinct waves propagate. First a tensile wave propagates at the speed of sound in the elastic material. Behind the longitudinal wavefront the elastic membrane is stretched. A transverse wave, accompanied by fluid motion, propagates in the stretched region. The transverse wave presents a dispersion relation similar to capillary waves but the equivalent "surface tension" is the tension in the membrane, which results from the impact. We investigate the coupling between the two waves, documenting the variation of the "surface tension" with impact speed. The deceleration dynamics of the impactor and the instability of the membrane giving rise to wrinkles will also be discussed.

3:07PM D32.00005 Marangoni forces in interfacial dilatational rheology , GWYNN ELFRING, GARY LEAL, TODD SQUIRES, University of California Santa Barbara — Many methods for measuring the mechanical properties of fluid interfaces involve generating a flow at the interface with both dilatation and shear, such as by translating a probe through a fluid interface. We examine here the force on a translating probe at an interface laden with a soluble surfactant, that exhibits Newtonian interfacial rheology. We assume that the interface is neither incompressible nor equilibrated. We look at the effects on the force measured by a probe due to small deviations in the concentration field which result from the dilatational flows induced by the probe. In particular we discuss the coupling of concentration relaxation and surface viscosity on Marangoni forces generated by the interfacial flows.

3:20PM D32.00006 ABSTRACT WITHDRAWN –

3:33PM D32.00007 Surfactant on a Thin Liquid Layer: Outward Spreading[1] , ELLEN SWANSON, Centre College, STEPHEN STRICKLAND, MICHAEL SHEARER, KAREN DANIELS, North Carolina State University — The spreading of insoluble surfactant molecules on a thin liquid layer can be modeled by a coupled system of two fourth order partial differential equations. The equations for the surfactant concentration and the height of the liquid layer are derived from the Stokes equations using the lubrication approximation, and have been accepted as an accurate prediction of the spreading behavior for over two decades. In experiments, we measure the surfactant concentration profile through fluorescence imaging of tagged lipids, while simultaneously measuring the height profile with laser profilometry. We compare the outward spreading observed in the experiment to that predicted by the numerical simulations of the model, and find inconsistencies for initial surfactant distributions both above and below the critical monolayer concentration. Similar inconsistencies are also found when the surfactant is spreading inward; these are described in the companion talk that follows.

[1]NSF DMS-0968258, RCSA 19788

3:46PM D32.00008 Surfactant on a Thin Liquid Layer: Self-Healing Dynamics[1] , STEPHEN STRICKLAND, North Carolina State University, CAMERON CONTI, MATT HIN, RICHARD SAYANAGI, Harvey Mudd College, KAREN DANIELS, North Carolina State University, RACHEL LEVY, Harvey Mudd College — As in the previous talk, we explore the dynamics of an insoluble surfactant spreading on a thin viscous Newtonian layer. Here, a central disk-shaped surfactant-free region heals, with the fluid layer ideally becoming entirely coated with surfactant. In the initial dynamics of this self-healing, Marangoni forces drive an axisymmetric annular ridge inward to coalesce into a growing central fluid distension, unlike outward spreading in which the ridge decays. In later dynamics, this distension slowly relaxes and the surfactant concentration equilibrates. We measure the surfactant concentration profile through fluorescence imaging of tagged lipids, while simultaneously measuring the height profile with laser profilometry. For surfactant concentrations close to but below the critical monolayer concentration, we observe agreement between the height profiles in the numerical simulations and the experiment, but disagreement in the surfactant distribution. In experiments at lower concentrations, the surfactant spreading and formation of a Marangoni ridge are not present, leaving a hole which is essentially surfactant-free. This observation, not captured at all in simulations, may have undesirable implications for applications such as drug delivery.

[1]NSF DMS-0968258, RCSA 19788

3:59PM D32.00009 On the Measurement of Longitudinal Interfacial Waves and Surfactant Dynamic Properties[1] , N. WASHUTA, X. LIU, University of Maryland, G.M. KORENOWSKI, Rensselaer Polytechnic Institute, J.H. DUNCAN, University of Maryland — The behavior of longitudinal interfacial waves at a surfactant-laden air-water interface is studied experimentally. The waves are generated in a glass tank (4.5 cm deep, 17.8 cm wide, and 75 cm long) by horizontal oscillation of a thin Teflon barrier, which spans the width of the tank. Local surfactant concentration is measured instantaneously and non-intrusively by using a nonlinear optical method called Second Harmonic Generation (SHG). In this method, a laser pulse with a wavelength of 532 nm is reflected off of the free surface at a 60-degree angle. The reflected beam contains both the original wavelength of 532 nm and its second harmonic, 266 nm. The ratio of the intensity of the reflected 266-nm signal to the 532-nm signal is proportional to the square of the local surfactant concentration at the interface. This SHG signal is recorded at a range of distances from the barrier for several barrier-oscillation frequencies and the resulting data is used to determine the wavelengths, phase speeds and amplitude decay rates of the longitudinal waves. Applying the linear theory of longitudinal waves, these wave propagation characteristics are then used to determine the dynamic properties of the surfactant.

[1]The support of the National Science Foundation, Division of Ocean Sciences is gratefully acknowledged.

4:12PM D32.00010 Marangoni flow induced by alcohol deposition on a water film , FEDERICO HERNANDEZ-SANCHEZ, ANTONIN EDDI, JACCO SNOEIJER, University of Twente — Bringing the interfaces of two miscible fluids into contact naturally generates strong gradients in surface tension. Here we investigate such a Marangoni-driven flow by continuously supplying isopropyl alcohol (IPA) on a film of water, using microdrops of IPA-water mixtures. These droplets create a localized depression in surface tension that leads to the opening of a circular hole in the water film, with water being collected in a growing rim at the edge of the hole. The dynamics of the thin film is monitored experimentally using high-speed imaging. We find that the radius of the hole opens as $r \sim t^{1/2}$. This result can be explained from a balance between Marangoni and viscous stresses, assuming that the gradients in surface tension are smoothened out over the entire size of the hole. We derive a scaling law that accurately predicts the influence of the IPA flux as well as the thickness of the thin film at the interior of the hole.

Sunday, November 24, 2013 2:15PM - 4:25PM –

Session D33 Drops III: Electric Field Effects 404 - Aditya Khair, Carnegie Mellon University

2:15PM D33.00001 Electrohydrodynamic interactions in Quincke rotation: from pair dynamics to collective motion , DEBASISH DAS, DAVID SAINTILLAN, University of Illinois, Urbana-Champaign — Weakly conducting dielectric particles suspended in a dielectric liquid can undergo spontaneous sustained rotation when placed in a sufficiently strong dc electric field. This phenomenon of Quincke rotation has interesting implications for the rheology of these suspensions whose effective viscosity can be reduced by application of an external field. While previous models based on the rotation of isolated particles have provided accurate estimates for this viscosity reduction in dilute suspensions discrepancies have been reported in more concentrated systems where particle-particle interactions are likely significant. Motivated by this observation we extend the classic description of Quincke rotation based on the Taylor-Melcher leaky dielectric model to account for pair electrohydrodynamic interactions between identical spheres using method of reflections. We also consider the case of spherical particles undergoing Quincke rotation next to a planar electrode, where hydrodynamic interactions with the no-slip boundary lead to a self-propelled velocity. The interactions between such Quincke rollers are analyzed, and a transition to collective motion is predicted in sufficiently dense collections of many rollers, in agreement with recent experiments.

2:28PM D33.00002 Electrorotation of a viscous droplet in a uniform direct current electric field[1] , HUI HE, PAUL SALIPANTE, PETIA VLAHOVSKA, Brown University — We study both analytically and numerically the experimentally observed nonaxisymmetric droplet deformation and orientation in a uniform DC electric field [1]. The theoretical model shows that above a threshold electric field a rotational flow is induced about the droplet. As a result, drop shape becomes a general ellipsoid with major axis obliquely oriented to the applied field direction. The theory is in excellent agreement with the experimental data for high viscosity drops [2]. Low viscosity drops undergo significant deformation during rotation, which is captured by numerical simulations using the boundary integral method.

[1] Salipante and Vlahovska, Physics of Fluids, 22:112110 (2010)
[2] He, Salipante, and Vlahovska, Physics of Fluids,25:032106 (2013)

[1]This work was supported by NSF-CBET award 1132614.

2:41PM D33.00003 The Influence of Inertia and Charge Relaxation on Electrohydrodynamic Drop Deformation , JAVIER LANAUZE, LYNN WALKER, ADITYA KHAIR, Dept. of Chemical Engineering, Carnegie Mellon University — An analytical model is presented for the electrohydrodynamic deformation of a drop in a leaky-dielectric fluid under a uniform electric field, taking into account transient fluid inertia and a finite electrical relaxation time. Capillary forces are assumed to be sufficiently large that the drop only slightly deviates from its equilibrium spherical shape. The temporal droplet deformation is governed by two dimensionless groups: (i) the ratio of capillary to momentum diffusion time scales: an Ohnesorge number Oh; and (ii) the ratio of charge relaxation to momentum diffusion time scales, which we denote by Sa. If charge and momentum relaxation occur quickly compared to interface deformation, $Sa \ll 1$ and $Oh \gg 1$ for the droplet and medium, a monotonic deformation is acquired. In contrast, $Sa > 1$ and $Oh < 1$ for either phase can lead to a non-monotonic deformation. In the latter case, the droplet and medium behave as perfect dielectrics at early times, which always favors an initial prolate deformation. Consequently, for a final oblate deformation, there is a shape transition from prolate to oblate at intermediate times. After the transition there may be an overshoot in the deformation, which is proceeded by an algebraic tail describing the arrival towards steady state.

2:54PM D33.00004 Deformation of leaky-dielectric fluid globules under strong electric fields: Boundary layers and jets at large Reynolds numbers , ORY SCHNITZER, ITZCHAK FRANKEL, EHUD YARIV, Technion - Israel Institue of Technology — In Taylor's theory of electrohydrodynamic drop deformation (Proc. R. Soc. Lond. A, vol. 291, 1966, pp. 159-166), inertia is neglected at the outset, resulting in fluid velocity that scales as the square of the applied-field magnitude. For large drops, with increasing field strength the Reynolds number predicted by this scaling may actually become large, suggesting the need for a complementary large-Reynolds-number investigation. Balancing viscous stresses and electrical shear forces in this limit reveals a different velocity scaling, with the 4/3-power of the applied-field magnitude. We focus here on the flow over a gas bubble. It is essentially confined to two boundary layers propagating from the poles to the equator, where they collide to form a radial jet. At leading order in the Capillary number, the bubble deforms due to (i) Maxwell stresses; (ii) the hydrodynamic boundary-layer pressure associated with centripetal acceleration; and (iii) the intense pressure distribution acting over the narrow equatorial deflection zone, appearing as a concentrated load. Remarkably, the unique flow topology and associated scalings allow to obtain a closed-form expression for this deformation through application of integral mass and momentum balances. On the bubble scale, the concentrated pressure load is manifested in the appearance of a non-smooth equatorial dimple.

3:07PM D33.00005 Retreating behavior of a charged ionic liquid droplet in a dielectric liquid under electric field[1] , MYUNG MO AHN, DO JIN IM, IN SEOK KANG, POSTECH, Pohang, Gyeongbuk, South Korea — Ionic liquids show great promise as excellent solvents or catalysts in energy and biological fields due to their unique chemical and physical properties. The ionic liquid droplets in microfluidic systems can also be used as a potential platform for chemical biological reactions. In order to control electrically the ionic liquid droplets in a microfluidic device, the charging characteristics of ionic liquid droplets need to be understood. In this work, the charging characteristics of various ionic liquids are investigated by using the parallel plate electrodes system. Under normal situation, a charged droplet shows bouncing motion between electrodes continuously. However, for some special ionic liquids, interesting retreating behavior of charged ionic liquid droplet has been observed. This retreating behavior of ionic liquid droplet has been analyzed experimentally by the image analysis and the electrometer signal analysis. Based on the hypothesis of charge leakage of the retreating ionic liquid droplets, FT-IR spectroscopy analysis has also been performed. The retreating behavior of ionic liquid droplet is discussed from the intermolecular point of view according to the species of ionic liquids.

[1]This research was supported by grant No. 2013R1A1A2011956 funded by the Ministry of Science, ICT and Future Planning (MSIP) and by grant No. 2013R1A1A2010483 funded by the Ministry of Education, Science and Technology (MEST) through the NRF.

3:20PM D33.00006 A numerical method for electro-kinetic flow with deformable fluid interfaces , MICHAEL BOOTY, New Jersey Institute of Technology, MANMAN MA, Shanghai Jiao Tong University, MICHAEL SIEGEL, New Jersey Institute of Technology — We consider two-phase flow of ionic fluids whose motion is driven by an imposed electric field. At a fluid interface, a screening cloud of ions develops and forms an electro-chemical double layer or Debye layer. The imposed field acts on this induced charge distribution, resulting in a strong slip flow near the interface. We formulate a "hybrid" or multiscale numerical method in the thin Debye layer limit that incorporates an asymptotic analysis of the electrostatic potential and fluid dynamics in the Debye layer into a boundary integral solution of the full moving boundary problem. Results of the method are presented that show time-dependent deformation and steady state drop interface shapes when the timescale for charge-up of the Debye layer is either much less than or comparable to the timescale of the flow.

3:33PM D33.00007 Electrohydrodynamics of drops covered with small particles , MALIKA OURIEMI, PETIA VLAHOVSKA, Brown University, School of Engineering — A weakly conductive drop immersed in a more conductive liquid first undergoes an oblate deformation, and then experiences a rotation similar to Quincke rotation when submitted to an increasing DC uniform electrical field. We present an experimental study of a drop with an interface partially or completely covered with microscopic particles. Depending on the field intensity, the surface coverage, and the characteristics of the particles, the drop exhibits: (i) prolate deformation, (ii) emergence of pattern of sustained particle motions, or (iii) decrease of the electrical field that induces rotation.

3:46PM D33.00008 Numerical simulations of a ferrofluid drop on a substrate under an applied magnetic field , IVANA SERIC, SHAHRIAR AFKHAMI, LOU KONDIC, New Jersey Institute of Technology — Understanding the behavior of a ferrofluid drop on a surface is of direct relevance to applications such as adaptive liquid lens optics. Here, we consider a ferrofluid drop on a horizontal substrate subjected to an applied uniform magnetic field with a non-magnetizable passive gas atop. Governing equations are the static Maxwell equations coupled with the incompressible Navier-Stokes equations. We use the long wave approximation to derive the equation that governs the non-linear evolution of the drop interface. Contact angles are imposed through the disjoining pressure model. The evolution equation is solved numerically and the results are compared with data from experiments.

3:59PM D33.00009 Electrowetting-driven spreading and jumping of drops in oil , JIWOO HONG, SANG JOON LEE, Department of Mechanical Engineering, Pohang University of Science and Technology (POSTECH) — Electrowetting-based practical applications include digital microfluidics, liquid lenses, and reflective displays. Most of them are performed in water/oil system, because oil medium reduces the contact-angle hysteresis and prevents drop evaporation. In this study, the effects of drop volume, oil viscosity, and applied voltage on the dynamic behaviors of spreading drops, such as transition of spreading pattern and response time, are investigated. Interestingly, jumping phenomena of drops are observed in oil when the applied voltage is turned off after reaching the electrowetted equilibrium radius of drops. A numerical model to predict the transient behavior of jumping drops is formulated based on the phase-field method. The numerical results for the transient deformation of jumping drops show quantitative agreement with the experimental results.

4:12PM D33.00010 Wetting of sessile water drop under an external electrical field , VALERIE VANCAUWENBERGHE, Aix-Marseille University, PAOLO DI MARCO, University of Pisa, DAVID BRUTIN, Aix-Marseille University, AMU COLLABORATION, UNIPI COLLABORATION — The enhancement of heat and mass transfer using a static electric field is an interesting process for industrial applications, due to its low energy consumption and potentially high level of evaporation rate enhancement. However, to date, this phenomenon is still not understood in the context of the evaporation of sessile drops. We previously synthesized the state of the art concerning the effect of an electric field on sessile drops with a focus on the change of contact angle and shape and the influence of the evaporation rate [1]. We present here the preliminary results of an new experiment set-up. The novelty of the set-up is the drop injection from the bottom that allows to generate safety the droplet under the electrostatic field. The evaporation at room temperature of water drops having three different volumes has been investigated under an electric field up to 10.5 kV/cm. The time evolutions of the contact angles, volumes and diameters have been analysed. As reported in the literature, the drop elongate along the direction of the electric field. Despite the hysteresis effect of the contact angle, the receding contact angle increases with the strength of the electric field. This is clearly observable for the small drops for which the gravity effect can be neglected.

Sunday, November 24, 2013 2:15PM - 4:25PM –

Session D34 Drops IV: Particle-Laden Drops 405 - Virgil Adumitroaie

2:15PM D34.00001 Impact of particle laden drops onto surfaces of various wettability , VIKTOR GRISHAEV, CARLO SAVERIO IORIO, Microgravity Research Centre, Université libre de Bruxelles, EP-CP165/62, B-1050, Brussels, Belgium, ALIDAD AMIRFAZLI, Department of Mechanical Engineering 425 LSB Bldg, York University, 4700 Keele St, Toronto, ON, M3J 1P3, Canada — Understanding characteristics of the impact of complex drops (liquid and solid particles) onto substrates is important for many applications, e.g. for additive manufacturing. Complex drops (diameter $\approx$ 3.8 mm) were produced using polyethylene microspheres dispersed in deionized water. Drop impact was investigated on substrates with different wettability (contact angles $<5^\circ$ and 95°) using high-speed cameras and image processing methods. By varying speed of the drops (1.7 to 3.7 m/s), diameter of the particles (200 and 500 μm) and particles volume fraction (0 to 35%), the influence of these variables on the impact dynamics of the drop and the distribution of particles after the impact was studied. For hydrophilic substrate, drops with 200 μm particles are arranged in a ring after the impact. For hydrophobic substrate, drops can split into several drops depending on drop velocity, size and volume fraction of particles. Also, the dynamics of drop spreading was elucidated.

2:28PM D34.00002 Yield stress fluid droplet impact on coated surfaces , BRENDAN BLACKWELL, MARC DEETJEN, RANDY EWOLDT, University of Illinois at Urbana-Champaign — Yield stress fluids, including gels and pastes, are effectively fluid at high stress and solid at low stress. In liquid-solid impacts, these fluids can stick and accumulate where they impact, motivating several applications of these rheologically-complex materials. Here we use high-speed imaging to experimentally study liquid-solid impact of yield stress fluids on dry and precoated surfaces. With a precoating of the same material, we can observe large, long-lifetime ejection sheets with redirected momentum that extend away from the impact location. Under critical splash conditions, sheet breakup occurs and ejected droplets can be nonspherical and threadlike due to the inability of capillary stresses to deform material above a certain lengthscale. By varying the droplet size, impact velocity, surface coating thickness, and rheological material properties, we develop appropriate dimensionless parameters, quantify splash characteristics including height and radial extent of deposition, and present a regime map of impact behaviors.

2:41PM D34.00003 Monolayer Splat , WENDY W. ZHANG, LUUK A. LUBBERS, QIN XU, SAM WILKEN, HEINRICH M. JAEGER, James Franck Institute and Department of Physics, The University of Chicago, Chicago, Illinois 60637, USA — We investigate experimentally and numerically the evolution of dense suspension drops that collide against a smooth solid surface and flatten into a rapidly expanding monolayer. Further expansion of the monolayer creates a lace-like pattern of particle clusters separated by particle-free regions. Agreement between the measured expansion rates, and those calculated using a force balance on the outermost particle, demonstrates that the splat expands as fast as the edge particle moves. We also present a model describing how the inhomogeneous particle distribution within the splat grows from slight initial fluctuations. Two-dimensional simulations using this model yield average instability growth rates and saturation values in quantitative agreement with measurements. This shows the spatial inhomogeneity can be captured by balancing forces acting on individual particles as well.

2:54PM D34.00004 Splashing onset in dense suspension droplets , IVO R. PETERS, MARTIN H. KLEIN SCHAARSBERG, QIN XU, HEINRICH M. JAEGER, University of Chicago — We investigate the splashing onset of droplets of dense suspensions upon the impact onto a solid substrate. Unlike in the splashing of pure liquid droplets, the ejecta of dense suspensions are individual solid particles. We show that a global hydrodynamic balance is unable to predict the splashing onset and propose to replace it by an energy balance at the level of the particles in the suspension. The key ingredient for this balance is to take into account collisions between the particles at the outer edge of the droplet. We experimentally verify that the resulting, particle-based Weber number gives a reliable, particle size and density dependent splash onset criterion. We further show that the same argument also explains why, in bimodal systems, smaller particles are more likely to escape than larger ones. Finally, we show that increasing the viscosity of the suspending liquid effectively decreases the efficiency of the collisions between particles and, consequently, increases the splash onset velocity.

3:07PM D34.00005 Dense suspension splash , KEVIN M. DODGE, IVO R. PETERS, JAKE ELLOWITZ, MARTIN H. KLEIN SCHAARSBERG[1], HEINRICH M. JAEGER, WENDY W. ZHANG, Physics Department & the James Franck Institute, University of Chicago — Upon impact onto a solid surface at several meters-per-second, a dense suspension plug splashes when liquid-coated particles are ejected from the plug bulk. We use discrete-particle simulations to examine the momentum transfers responsible for splash formation. We find that a simulation using a densely packed plug containing dry, noncohesive, inelastic grains reproduces the unexpected experimental finding that mixing larger particles into a suspension of small particles creates a bigger splash [1]. We also find that with increasing impact speed, the measured momentum distribution of ejected particles tends toward the result from the dry-grains-only simulation. These results support the idea that the splash from a low-viscosity solvent suspension is created by inertia-dominated collisions between particles. In this regime, viscous drag from the interstitial fluid is negligible. In future work, we will examine splash formation in simulations where particles approaching contact do experience viscous drag.

[1] Peters et al, Phys. Rev. Lett. **111**, 028301 (2013).

[1]Current Address: Physics of Fluids Group, University of Twente, P. O. Box 217, 7500 AE Enschede, The Netherlands

3:20PM D34.00006 Electrostatic control of the coffee stain effect[1] , ALEX WRAY, DEMETRIOS PAPAGEORGIOU, Imperial College London, KHELLIL SEFIANE, University of Edinburgh, OMAR MATAR, Imperial College London — The "coffee stain effect," as first explained by Deegan et al. 1997, has received a great deal of attention amongst modellers and experimentalists in recent years, perhaps due in part to its obvious casual familiarity. However, it maintains interest because of its intriguing reliance on an interplay of a trio of effects: contact line pinning, inhomogeneous mass flux, and resulting capillarity-driven flow. What is more, the effect, and especially its suppression or reversal, find applications in fields as diverse as sample recovery, mass spectroscopy and the printing of Organic LEDs. We examine the motion a nanoparticle-laden droplet deposited on a precursor film, incorporating the effects of capillarity, concentration-dependent rheology, together with a heated substrate and resultant mass flux and Marangoni effects. We allow the substrate to act as an electrode and incorporate a second electrode above the droplet. The potential difference together with a disparity in electrical properties between the two regions results in electrical (Maxwell) stresses at the interface. We show via lubrication theory and via direct numerical simulations that the ring effect typically observed may be suppressed or augmented via appropriate use of electric fields.

[1]EPSRC DTG

3:33PM D34.00007 Spontaneous formation of nanostructures inside inkjet-printed colloidal drops , XIN YANG, NATHANIEL THORNE, YING SUN, Department of Mechanical Engineering and Mechanics, Drexel University — Nanostructures formed in inkjet-printed colloidal drops are systematically examined with different substrates and ink formulations. Various deposition patterns from multi-ring, radial spoke, firework to spider web, foam and island structures are observed. With a high particle loading, deposition transitions from multi-ring near the drop edge to spider web and finally to foam and islands in the center of the drop with 20 nm sulfate-modified polystyrene particles. At the same particle loading, 200 nm particles self-assemble into radial spokes at the drop edge and islands in the center, due to reduced contact line pinning resulted from less particles. In drops with a low particle concentration, due to fingering instability of the contact line, 20 nm particles form radial spokes enclosed by a ring, while 200 nm particles assemble into firework-like structures without a ring. Moreover, at a high particle loading, ruptures are observed on the multi-ring structure formed by 20 nm carboxylic-modified particles, due to stronger capillary forces from the contact line. Furthermore, for a drop printed on a less hydrophilic substrate, the interparticle interactions enable a more uniform deposition rather than complex nanostructures.

3:46PM D34.00008 Building micro-soccer-balls with evaporating colloidal fakir drops , HANNEKE GELDERBLOM, ÁLVARO G. MARÍN, Physics of Fluids, University of Twente, The Netherlands, ARTURO SUSARREY-ARCE, ARIE VAN HOUSSELT, LEON LEFFERTS, Catalytic Processes and Materials, University of Twente, The Netherlands, HAN GARDENIERS, Mesoscale Chemical Systems, University of Twente, The Netherlands, DETLEF LOHSE, JACCO H. SNOEIJER, Physics of Fluids, University of Twente, The Netherlands — Drop evaporation can be used to self-assemble particles into three-dimensional microstructures on a scale where direct manipulation is impossible. We present a unique method to create highly-ordered colloidal microstructures in which we can control the amount of particles and their packing fraction. To this end, we evaporate colloidal dispersion drops from a special type of superhydrophobic microstructured surface, on which the drop remains in Cassie-Baxter state during the entire evaporative process. The remainders of the drop consist of a massive spherical cluster of the microspheres, with diameters ranging from a few tens up to several hundreds of microns. We present scaling arguments to show how the final particle packing fraction of these balls depends on the drop evaporation dynamics, particle size, and number of particles in the system.

3:59PM D34.00009 Influence of Relative Humidity on the Spreading Dynamics of a Drying Drop of Whole Blood , WASSIM BOU ZEID, DAVID BRUTIN, Aix-Marseille University — Newtonian and non-Newtonian fluids start spreading after coming into contact with a solid substrate till the anchoring of the triple line. Our experimental work aims to study the effect of the relative humidity on the spreading dynamics of drops of whole blood. Drops of blood of same volume (+/- 4.8 %) are injected using a digital micropipette and gently deposited onto microscope ultraclean glass substrates. Experiments are conducted in a glove box at ambient temperature and a range of investigated relative humidities between 13.0% and 80.0%. The recorded images are post-processed using ImageJ in which the position of the contact line is measured every 20 ms. We show that the spreading dynamics is, in a first time, governed by relative humidity and later no more influence by relative humidity. Two spreading regimes are observed and analyzed compared to classical viscous drops. In previous work, we show that relative humidity influences the contact angle and the initial wetting radius. In the first regime, we find a spreading power law exponent that decreases for an increasing relative humidity. In the second regime, all the data collapse on each other and the evolution of the dimensionless radius no more depend on relative humidity.

4:12PM D34.00010 Opportunities for Fluid Dynamics Research in the Forensic Discipline of Bloodstain Pattern Analysis , DANIEL ATTINGER, Iowa State University, CRAIG MOORE, Niagara Regional Police Service, ADAM DONALDSON, Dalhousie University, ARIAN JAFARI, Iowa State University, HOWARD STONE, Princeton University — This review [Forensic Science International, vol. 231, pp. 375-396, 2013] highlights research opportunities for fluid dynamics (FD) studies related to the forensic discipline of bloodstain pattern analysis (BPA). The need for better integrating FD and BPA is mentioned in a 2009 report by the US National Research Council, entitled "Strengthening Forensic Science in the United States: A Path Forward". BPA aims for practical answers to specific questions of the kind: "How did a bloodletting incident happen?" FD, on the other hand, aims to quantitatively describe the transport of fluids and the related causes, with general equations. BPA typically solves the indirect problem of inspecting stains in a crime scene to infer the most probable bloodletting incident that produced these patterns. FD typically defines the initial and boundary conditions of a fluid system and from there describe how the system evolves in time and space, most often in a deterministic manner. We review four topics in BPA with strong connections to FD: the generation of drops, their flight, their impact and the formation of stains. Future research on these topics would deliver new quantitative tools and methods for BPA, and present new multiphase flow problems for FD.

Sunday, November 24, 2013 2:15PM - 4:12PM –

Session D35 Suspensions II: Fluid-Particle Interactions 406 -

2:15PM D35.00001 On the motion of a neutrally buoyant cylinder in simple shear flow , TSORNG-WHAY PAN, Department of Mathematics, University of Houston, Houston, TX 77204, SHIH-DI CHEN, SHIH-LIN HUANG, CHIN-CHOU CHU, CHIEN-CHENG CHANG, Institute of Applied Mechanics, National Taiwan University, Taipei 106, Taiwan, ROC — We have investigated the motion of a neutrally buoyant (NB) cylinder of circular or elliptic shape in simple shear flow by direct numerical simulation. The numerical results are validated by the comparisons with existing theoretical, experimental and numerical results. When placing a NB cylinder of either shape away from the centerline initially, it may migrate to an equilibrium position between the centerline and the wall, not the centerline, depending on the particle Reynolds number. Unlike the circular cylinder, the elliptic shape cylinder can migrate toward the without rotating at the higher particle Reynolds number due to the balanced torque from two sides.

2:28PM D35.00002 Rotation of porous ellipsoids in simple shear flows[1] , HASSAN MASOUD, Courant Institute of Mathematical Sciences, NYU and Department of Mechanical and Aerospace Engineering, Princeton University, HOWARD A. STONE, Department of Mechanical and Aerospace Engineering, Princeton University, MICHAEL J. SHELLEY, Courant Institute of Mathematical Sciences, NYU — We study theoretically the dynamics of porous ellipsoids rotating in simple shear flows. We use the Brinkman-Debye-Bueche (BDB) model to simulate flow within and through particles and solve the coupled Stokes-BDB equations to calculate the overall flow field and the rotation rate of porous ellipsoids. Our results show that the permeability has little effect on the rotational behavior of particles, and that the Jeffery's prediction of the angular velocity of impermeable ellipsoids in simple shear flows remains an excellent approximation, if not an exact one, for porous ellipsoids. We also examine the orientational diffusion of permeable ellipses and spheroids in the absence of a background flow. Employing an appropriate scaling, we present approximate expressions for the orientational diffusion of ellipses and spheroids. Our findings can serve as basis for developing a suspension theory for non-spherical porous particles.

[1]The support of the U.S. National Science Foundation Grant DMR-0844115 and the Institute for Complex Adaptive Matter is acknowledged.

2:41PM D35.00003 Dynamics of a Janus droplet in a linear shear flow , MISAEL DÍAZ-MALDONADO, Department of Chemical Engineering, University of Puerto Rico Mayagüez Campus, ANDREY IVANTSOV, SERGEY SHKLYAEV, Institute of Continuous Media Mechanics, Ural Branch of the Russian Academy of Sciences, UBALDO M. CÓRDOVA-FIGUEROA, Department of Chemical Engineering, University of Puerto Rico Mayagüez Campus — Janus droplet (JD) is an object with promising applications in the design of smart materials, microfluidics, drug delivery, etc. Despite numerous experimental works, theoretical aspects of JD dynamics remain almost unstudied. Our recent paper [Phys. Fluids, 2013 (accept. for publ.)] was devoted to the generalization of the classic Hadamard–Rybczynski problem, a flow past a droplet in a constant flow. However, in most applications, a JD is subject to a nonuniform flow; the simplest case of such a flow is linear. A perfect JD – a combination of two hemispherical domains – is considered; the interfaces are assumed nondeformable. In this case semianalytical solution is valid in terms of series with respect to Lamb's functions. First, we study the rotation of JD around the axis belonging to the internal interface and couple the angular velocity of the internal interface with the viscous torque imposed on JD. This problem, in particular, allows calculating a characteristic time of JD turn under an external torque. Then, the dynamics of JD in a 1D shear flow is analyzed. For arbitrary orientation of JD with respect to the external velocity field and its gradient, the problem is decomposed into five primitive problems. The force and torque for each of these cases are found.

2:54PM D35.00004 Numerical simulation of Stokes flow around particles via a hybrid Finite Difference-Boundary Integral scheme , AMITABH BHATTACHARYA, Indian Institute of Technology, Bombay, Mumbai — An efficient algorithm for simulating Stokes flow around particles is presented here, in which a second order Finite Difference method (FDM) is coupled to a Boundary Integral method (BIM). This method utilizes the strong points of FDM (i.e. localized stencil) and BIM (i.e. accurate representation of particle surface). Specifically, in each iteration, the flow field away from the particles is solved on a Cartesian FDM grid, while the traction on the particle surface (given the the velocity of the particle) is solved using BIM. The two schemes are coupled by matching the solution in an intermediate region between the particle and surrounding fluid. We validate this method by solving for flow around an array of cylinders, and find good agreement with Hasimoto's (J. Fluid Mech. 1959) analytical results.

3:07PM D35.00005 Shear-Induced Diffusion of Cubic Colloids , STEVEN HUDSON, JOHN ROYER, NIST, DANIEL BLAIR, Georgetown University — Particles in many industrially relevant fluid suspensions have directional or anisotropic interactions, yet it is not understood how these interactions influence particle self-association or the rheology of a suspension. We therefore use confocal rheometry to study simultaneously the micro-scale particle motion and macro-scale rheology of a model colloidal suspension. Specifically, we study mono-disperse, hollow, silica cubes exhibiting well-characterized, well-controlled and tunable directional interactions. Tracking the 3-D position and orientation of the cubes as they move under steady shear, we characterize the packing structure and shear-induced diffusion of the cubes varying the shear rate, packing density, and depletion-induced attraction.

3:20PM D35.00006 Shear-induced diffusion of non-Brownian suspensions using a colored noise Fokker-Planck equation[1] , LAURA LUKASSEN, MARTIN OBERLACK, Chair of Fluid Dynamics, TU Darmstadt / Graduate School of Excellence Computational Engineering — In the Literature, shear-induced diffusion resulting from hydrodynamic interactions between particles, is described as a long-time diffusion. In contrast to the well-known Brownian diffusion which is described by a white noise force, several authors report that the former type of diffusion exhibits the particularity of a much longer correlation time of velocities. Further, Fokker-Planck equations describing this process of shear-induced diffusion have mostly been derived in position space. We present a considerably extended framework of the shear-induced diffusion problem, which essentially relies on the Markov process assumption under the consideration of long correlation times. Applying the mathematical machinery of Markov processes and Fokker-Planck equations, we conclude that this process may only be properly modelled by a Fokker-Planck approach if written in both position and velocity space. With this complementation we observe, that the long correlation times enter as a colored noise velocity. As a result, the Fokker-Planck equation also needs to be extended and we derive the Fokker-Planck equation for the shear-induced diffusion problem following the definitions of a colored noise Fokker-Planck equation.

[1]Graduate School of Excellence Computational Engineering

3:33PM D35.00007 Particle drifts in semi-dilute suspensions of highly viscous droplets , HUGUES BODIGUEL, FLORINDA SCHEMBRI, VINCENT MANSARD, ANNIE COLIN, Univ. Bordeaux — Though already the focus of many experimental and theoretical work, the origin and features of particle migration of semi-dilute suspensions is still in debate. Shear induced cross-stream migration is emphasized in microfluidic flows where high gradients of shear rate are obtained. We study suspensions made of highly viscous droplets in an index matched liquid as a model system. Particle deformation could be neglected, similarly to contact forces that are thought to play a role in suspensions of solid particles. In this work, we focus on a feature of particle migration which has been scarcely described, the particle drift in the flow direction. For that purpose, we developed a technique based on fluorescence photobleaching which enables us to measure simultaneously the particle and suspending fluid velocities. Particles are immersed in a solution containing fluorescein. Thanks to confocal microscopy, we follow the displacement of a bleached line together with the displacement of the particles. The results show that the particle velocity is generally lower than that of the suspending fluid, in a wide range of concentrations from 5 to 40%. Besides, we also observe a cross-stream migration that is quantified thanks to a balance with buoyancy and compared to existing theories.

3:46PM D35.00008 Arbitrary Lagrangian-Eulerian simulations of particle and bubble dynamics in non-Newtonian fluids , PENGTAO YUE, Virginia Tech — Fluid rheology affects particle-bubble interaction in various ways. For example, it modifies the migration of a single particle and a single bubble as well as the film drainage when they get close. In this talk, we will investigate these non-Newtonian effects using an arbitrary Lagrangian-Eulerian method which simultaneously tracks rigid particle surfaces and deformable bubble surfaces. The gas motion inside each bubble is neglected, and we only consider the bubble pressure which is determined by the isothermal ideal gas law. The particle motion and the fluid motion are solved in a unified Galerkin finite-element framework, in which the hydrodynamic forces and moments between the particle and the surrounding fluid cancel out. Mesh refinement is enforced where the surface curvature is high and where two boundary segments are close; the latter guarantees a sufficient resolution of the film drainage process. Numerical results on bubble migration and particle-bubble interaction in viscoelastic fluids and shear-thinning fluids will be presented.

3:59PM D35.00009 Rheological properties of suspensions of bubbles in a yield stress fluid , LUCIE DUCLOUE, GUILLAUME OVARLEZ, XAVIER CHATEAU, OLIVIER PITOIS, JULIE GOYON, Universite Paris-Est, Laboratoire Navier — We study the macroscopic response under shear of suspensions of bubbles in yield stress fluids. Model suspensions are prepared by mixing a monodisperse foam with a concentrated oil in water emulsion, both having the same continuous phase of a surfactant solution. The interstitial concentrated emulsion behaves as a solid viscoelastic material below a critical stress, and as a shear-thinning fluid above this yield stress. We measure the change in the macroscopic response (elastic modulus, yield stress, non-linear viscosity) due to the addition of bubbles to the fluid. We find that for a given emulsion, the elastic modulus is a decreasing function of the gas volume fraction ϕ, this decrease being all the sharper as the bubbles are big. We also observe that the yield stress of most studied materials is not modified by the presence of bubbles, whereas the non-linear viscosity during flow increases with ϕ. We show that those apparently contradictory changes in the behaviour are ruled by the deformability of the bubbles in the fluid. To quantify this effect, we introduce capillary numbers which compare the stresses exerted on a bubble during a measurement to the stresses due to surface tension. We thus compute an elastic capillary number in the solid regime, a plastic capillary number at the yield stress and a viscous capillary number during flow. Those numbers are very different in the solid and in the liquid regimes, explaining why the elastic, plastic and viscous properties do not follow the same evolution. Our results are quantitatively well predicted by a micromechanical approach.

Sunday, November 24, 2013 2:15PM - 4:25PM –

Session D36 Instability: General I 407 - Cedric Beaume, University of California, Berkeley

2:15PM D36.00001 Stability analysis of an impacting T-junction pipe flow[1] , KEVIN CHEN, CLARENCE ROWLEY, HOWARD STONE, Princeton University — The flow through a T-shaped pipe bifurcation (with the inlet at the bottom of the "T") is a common occurrence in both natural and man-made systems, including blood vessels, industrial pipe networks, and microfluidic channels. Despite the ubiquitous nature of the geometry, many questions about the flow physics remain. We analyze the stability of Navier–Stokes equilibria by executing numerical continuation on the Reynolds number (based on the average inlet velocity), using a combination of linear extrapolation and the Newton–GMRES algorithm. We find that the qualitative nature of the equilibria's local bifurcations is highly sensitive to the grid resolution. On a sufficiently resolved grid, a rapid succession of supercritical Hopf bifurcations begins at $Re \approx 550$. Visualizations of the neutrally stable eigenmodes reveal the physical nature of the instabilities. We also compare equilibria computed with different radii of curvature at the square corners of the "T." Next, a wavemaker analysis reveals the locations in the T-junction where the stability is most sensitive to localized changes in the dynamics, e.g., via a change in geometry.

[1]This work was supported by the NSF GRFP.

2:28PM D36.00002 Tomographic PIV Observations of the Growth of Localized Perturbations in Transitional Taylor-Couette Flow[1] , DANIEL BORRERO, MICHAEL SCHATZ, Georgia Institute of Technology — The flow between concentric rotating cylinders has been extensively studied over the years. Most studies have focused on the flow patterns that emerge from centrifugal instabilities and at highly turbulent regimes. More recently, however, there has been renewed interest in centrifugally stable Taylor-Couette flows, which bypass linear instability mechanisms and undergo a direct transition to turbulence. This transition shares many features with the direct transition to turbulence in other canonical shear flows that are linearly stable, such as pipe and plane Couette flows, including spatiotemporal intermittency and the coexistence of laminar and turbulent domains. We present tomographic PIV and flow visualization measurements of the growth of finite-size perturbations to the laminar state as they grow into persistent turbulent spots. In particular, we look at how the amplitude and duration of the perturbations affect the transition to turbulence and study the detailed three-dimensional structure of turbulent spots.

[1]This work was conducted with support from the National Science Foundation (CBET-0853691).

2:41PM D36.00003 Low-drag exact coherent states in Newtonian channel flow[1] , JAE SUNG PARK, MICHAEL GRAHAM, University of Wisconsin-Madison — Exact coherent states have been known to nicely capture the main features of turbulent flows such as near-wall coherent structures and streak spacing. In this study, we numerically calculate new classes of exact coherent states, specifically nonlinear traveling wave solutions, for Newtonian channel flow, which display low-drag flow features such as weak streamwise vortices and nearly nonexistent streamwise variations like those observed in polymer solutions and in Newtonian hibernating turbulence. Traveling wave solutions with various symmetries are found. While some of the structures clearly display nonlinear critical layer dynamics, in others this connection is not as clear. Dynamical trajectories are computed and some of the solutions are shown to lie on the basin boundary between laminar and turbulent flows and are thus edge-states of the flow. Lastly, the dependence of Reynolds number for the solutions is investigated. We find one intriguing family whose mean velocity profile appears to approach the so-called maximum drag reduction asymptote found in polymer solutions, despite the fact that fluid studied here is Newtonian. Our results suggest that these traveling wave solutions may play a role as promising targets for turbulence control strategies for drag reduction.

[1]This work was supported by the Air Force Office of Scientific Research through grant FA9550-11-1-0094 (Flow Interactions and Control Program).

2:54PM D36.00004 Exact near-wall traveling waves of plane Poiseuille flow , JOHN GIBSON, EVAN BRAND, University of New Hampshire — We present several spatially-localized equilibrium and traveling-wave solutions of plane Couette and plane Poiseuille flow. The solutions consist of highly concentrated and spanwise-localized alternating streamwise rolls, centered over low-speed streamwise streaks and flanked on either side by high-speed streaks. For large Reynolds numbers the solutions develop critical layers that are concentrated at isolated points on the critical surface $u = c$. For several traveling-wave solutions of plane Poiseuille flow, the rolls are concentrated near one wall, producing streaks near the wall and larger reduction of the bulk flow in the core. These solutions form particularly isolated and elemental versions of near-wall coherent structures in shear flows and capture, as precise time-independent solutions of Navier-Stokes, the process by which near-wall rolls exchange momentum between the wall and core regions and thereby increase drag.

3:07PM D36.00005 A doubly-localized solution of plane Couette flow , EVAN BRAND, JOHN GIBSON, University of New Hampshire — We present a new equilibrium solution of plane Couette flow localized in two spatially extended directions. The solution is derived from the EQ7/HVS solution of plane Couette flow discovered independently by Itano and Generalis (PRL 2009) and Gibson et al (JFM 2009), of which a spanwise localized version has also recently been produced (Gibson, these proceedings). The doubly localized solution displays relatively long length scales in comparison with the spatially periodic and spanwise localized solutions, suggesting the importance of these scales in capturing the spatial complexity of transitional and low-Reynolds number turbulence. The solution is comparable in size and appearance to the doubly-localized, chaotically evolving edge states previously computed in this flow by Duguet et al (PoF 2009) and Schneider et al (JFM 2010). Additionally, we address the structure of localized solutions in the "tails," i.e. in the region approaching laminar.

3:20PM D36.00006 Linear stability analysis of swirling turbulent flows with turbulence models[1] , VIKRANT GUPTA, MATTHEW JUNIPER, University of Cambridge — In this paper, we consider the growth of large scale coherent structures in turbulent flows by performing linear stability analysis around a mean flow. Turbulent flows are characterized by fine-scale stochastic perturbations. The momentum transfer caused by these perturbations affects the development of larger structures. Therefore, in a linear stability analysis, it is important to include the perturbations' influence. One way to do this is to include a turbulence model in the stability analysis. This is done in the literature by using eddy viscosity models (EVMs), which are first order turbulence models. We extend this approach by using second order turbulence models, in this case explicit algebraic Reynolds stress models (EARSMs). EARSMs are more versatile than EVMs, in that they can be applied to a wider range of flows, and could also be more accurate. We verify our EARSM-based analysis by applying it to a channel flow and then comparing the results with those from an EVM-based analysis. We then apply the EARSM-based stability analysis to swirling pipe flows and Taylor-Couette flows, which demonstrates the main benefit of EARSM-based analysis.

[1]This project is supported by EPSRC and Rolls-Royce through a Dorothy Hodgkin Research Fellowship.

3:33PM D36.00007 Electrokinetic Instability in Plane Poiseuille Flow , LUKAS VERMACH, Cambridge Centre for Analysis, University of Cambridge, C.P. CAULFIELD, BP Institute & DAMTP, University of Cambridge — We consider the linear stability of the flow of an electrically charged liquid driven by a constant pressure gradient through a plane channel with charged walls. The flow is modified by the establishment of electric double layers in the near-wall regions. Chakraborty & Das (2008 *Phys. Rev. E.* **77** 037303) introduced an extended theoretical model of the associated electroviscous effects, including the streaming field contribution produced as a result of the downstream motion of the charge carriers. We use this model to examine the impact of the streaming field on the background plane Poiseuille flow profile and hence the linear stability of the flow. We find that, under certain realistic circumstances involving sufficiently large surface potential, the streaming field strongly modifies the background flow, inducing inflection points in the velocity profile and near-wall reverse flow. We show that the critical Reynolds numbers for linear instability of such flows are independent of Péclet number, and can be substantially suppressed below that of the uncharged classical parabolic flow profile.

3:46PM D36.00008 Increasing lifetimes and the growing saddle of shear flow turbulence , TOBIAS M. SCHNEIDER, Max Planck Institute for Dynamics and Self-Organization, BRUNO ECKHARDT, Fachbereich Physik, Philipps-Universitaet Marburg, TOBIAS KREILOS, Max Planck Institute for Dynamics and Self-Organization & Fachbereich Physik, Philipps-Universitaet Marburg — In linearly stable shear flows turbulence spontaneously decays on a characteristic transient lifetime. The lifetime sharply increases with Reynolds number so that a possible divergence marking the transition to sustained turbulence at a critical point has been discussed, yet the mechanism underlying the increase has not been understood. We present a mechanism by which the lifetimes increase: a locally attracting orbit forms a "turbulent bubble" via a route-to-chaos sequence of bifurcations, followed by a boundary crisis in which the chaotic attractor turns into a chaotic saddle. The complexity of the turbulence supporting saddle hence increases and it becomes more densely filled with unstable periodic orbits, increasing the time it takes for a trajectory to leave the saddle and decay to the laminar state. We demonstrate this phenomenon in the state space of plane Couette flow and show that characteristic lifetimes vary non-smoothly and non-monotonically with Reynolds number.

3:59PM D36.00009 Flow Instability and Secondary Vortex Evolution in 90 Degree Bend , LIN NIU[1], HUA-SHU DOU[2], Zhejiang Sci-Tech University, FLUID MECHANICS RESEARCH TEAM — Three-dimensional incompressible Navier-Stokes equations are employed to simulate the laminar flow in a 90 degree bend with square cross-section. Then, the energy gradient theory is used to analyze the stability of the flow. The Reynolds number based on the channel width and the averaged velocity is 158, 394 and 790, respectively. It is found that at Re=790, the value of the energy gradient function K increases as the fluid entering the curved section, causing flow instability and forming a pair of secondary vortices; then the secondary vortices gradually stabilizes and the value of K decreases. At the exit of the bend, the total pressure distribution in the cross-section presents serious distortion, which leads to a peak of K. As such, it promotes instability of the flow and causes a transition of two vortices to four vortices. With the flow ahead, the maximum of K in the cross section rises again, resulting in the transition of four vortices to eight vortices. While at low Re (Re = 158 and Re = 394), there is only one pair of vortices in the bend, which are stable, due to low value of K. This study shows that the occurrence of instability is closely related to the evolution of energy gradient function K.

[1]Graduate student in Fluid Mechanics.
[2]Professor in Fluid Mechanics; AIAA Associate Fellow.

4:12PM D36.00010 Kelvin-Helmholtz instabilities and Bénard Von-Karman Streets under lateral confinement , LUC LEBON, PAUL BONIFACE, MATHIEU RECEVEUR, LAURENT LIMAT, Matiere et Systemes Complexes (MSC), CNRS and Univ Paris Diderot, Paris, France, FABIEN BOUILLET, Saint-Gobain Recherches, Aubervilliers, France — We have investigated Kelvin-Helmholtz instabilities in a confined geometry. We used a large tank of water with a belt moving at high speed on the central part of its free surface. The water below the belt is dragged by this one, while the excess is recirculating along the lateral walls. Using displaceable walls, belts of different widths, and modifying the water height, it is possible to tune at will the geometry. Depending on the involved ratios, two different behaviors are observed: (1) recirculation by the bottom of the tank, (2) recirculation along the walls with the growth of two coupled Kelvin-Helmholtz instabilities on each side of the belt. At long time scale, and depending again on the involved geometry, the flow evolves to a 3D turbulence or to a well organized Bénard-Von-Karman street, with a 2D spatial organization of the flow. The wave-length in each vortex row is in agreement with a stability calculations of point vortices developed in the 30's by Rosenhead.

Sunday, November 24, 2013 4:45PM - 5:50PM –

Session E1 Geophysical: Oceanographic III 323 - Eric Paterson, Virginia Polytechnic Institute and State University

4:45PM E1.00001 On the study of radar backscattering of ocean surface in response to rainfall[1] , XINAN LIU, QUANAN ZHENG, REN LIU, JAMES H. DUNCAN, University of Maryland, College Park — A model of radar backscattering from the ocean surface in response to rainfall is developed. The model shows that the radar return intensity is a function of the wavelength and incident angle of the radar waves and the rain rate. The model explains the differences between the radar response to rain rate simultaneously observed by C-band ASAR and ground-based weather radar. An experiment on the simultaneous measurements of the characteristics of the ocean surface in response to rainfall and its radar back-scatter is performed in the laboratory. The experiment is carried out in a water pool that is 1.22 m by 1.22 m with a water depth of 0.3 m. Artificial rainfall is generated from an array of hypodermic needles. The surface characteristics including crowns, stalks and ring waves are measured with a cinematic Laser-Induced-Florescence (LIF) technique while secondary droplets are measured with a shadowgraph technique. The radar backscattering signal is recorded with a dual-polarized, ultra-wide band radar. The frequency dependence and polarization of the radar signatures due to the surface features are discussed.

[1]The work is supported by National Science Foundations, Division of Ocean Science.

4:58PM E1.00002 Turbulent bubbly flow under unsteady breaking waves[1] , MORTEZA DERAKHTI, JAMES KIRBY, Department of Civil and Environmental Engineering, University of Delaware — Wave breaking is a highly dissipative process, and also a source of turbulence in the ocean surface layer. It entrains a large volume of air in bubbles that rapidly evolves into a distribution of bubble sizes which interacts with fluid turbulence and organized motions, leads to a continuum time-dependent void fraction of bubbles. In this presentation, we use a 3D VOF-based Navier-Stokes solver extended to incorporate entrained bubble populations using an Eulerian-Eulerian formulation for a poly-disperse bubble phase, to consider an isolated, deep-water breaking event. We examine anisotropic non-stationary turbulence structure, momentum exchange between dispersed bubbles and liquid phase, bubble effects on mean and turbulent field, shear- and bubble-induced dissipation, bubble void fraction distribution and integral properties of the bubble plume both in spilling and plunging breakers. Comparison of mean and turbulent velocities, void fraction distributions and integral properties of the bubble plume show that the model is capable of capturing the large scale of turbulence and bubble plume kinematics and dynamics fairly well, and the inclusion of bubbles gives better results in terms of total dissipation and turbulent velocities.

[1]This study was supported by the office of Naval Research, Littoral Geosciences and Optics Program, grants N00014-10-1-0088 and N00014-13-1-0124.

5:11PM E1.00003 Magnetic Field Induced by a Submerged Inhomogenous Current , DANIEL SOBIEN[1], ERIC PATERSON[2], Virginia Tech, Aerospace and Ocean Engineering — A one-way coupled approach has been developed for studying the electromagnetic field induced by a submerged inhomogenous current. The method is based upon solving the Navier-Stokes equations, transport equations for salinity and temperature, the UNESCO equation of state for seawater density and conductivity, and a steady-state Poisson equation for the magnetic-field perturbation. The computational domain includes both the ocean and the atmosphere. Simulations are conducted for a net-zero-momentum wake in a linearly stratified ocean, and the influence of depth and Brunt-Väisälä frequency is studied. Simulation data will quantify the magnitude and distribution of magnetic-field perturbation.

[1]Graduate Research Associate
[2]Professor and Department Head

5:24PM E1.00004 Large-eddy simulation of oil slicks from deep water blowouts[1] , DI YANG, Department of Mechanical Engineering, Johns Hopkins University, MARCELO CHAMECKI, Department of of Meteorology, The Pennsylvania State University, CHARLES MENEVEAU, Department of Mechanical Engineering, Johns Hopkins University — Deep water blowouts generate plumes of oil droplets and gas bubbles that rise through, and interact with various layers of the ocean. When plumes reach the ocean mixed layer (OML), the interactions among plume, Ekman Spiral and Langmuir turbulence strongly affect the final rates of dilution and bio-degradation. The present study aims at developing a large-eddy simulation (LES) capability for the study of the physical distribution and dispersion of petroleum (oil and gas) under the action of physical oceanographic processes in the OML. In the current LES, the velocity and temperature fields are simulated using a hybrid pseudo-spectral and finite-difference scheme; the oil/gas field is described by an Eulerian concentration field and it is simulated using a bounded finite-volume scheme. A variety of subgrid-scale models for the flow solver are implemented and tested. The LES capability is then applied to the simulation of oil plume dispersion in the OML, which is initially released from a point source below the thermocline. Graphical visualization of the LES results shows surface oil slick distribution consistent with the satellite and aerial images of surface oil slicks reported in the literature.

[1]Funding from the GoMRI RFP-II is gratefully acknowledged.

5:37PM E1.00005 Comparing fixed and dynamic-salinity models of sea ice , DAVID REES JONES, GRAE WORSTER, University of Cambridge — The bulk salinity of sea ice has long been poorly represented in climate models. We have incorporated a physically-derived parameterization of ice desalination by gravity drainage in terms of a convective upwelling velocity into a one-dimensional thermodynamic sea-ice model of the kind currently used in coupled climate models. Our parameterization allows us to determine salt fluxes from sea ice corresponding to the evolution of the bulk salinity of the ice, in contrast to current, established models that prescribe the ice salinity. This improves the predictive power and responsiveness of climate models in terms of buoyancy fluxes to the polar oceans, and also the thermal and mechanical properties of sea ice, which depend on its salinity. We discuss and compare our parameterization to other recent parameterizations of gravity drainage, as well as existing fixed-salinity models, both in terms of laboratory experiments and deep ocean mixed layer calculations in the case of growing first-year ice. These comparisons explain why the direct effect of ice salinity on growth is relatively small (though not negligible), and highlight substantial differences in salt fluxes into the polar oceans.

Sunday, November 24, 2013 4:45PM - 5:50PM –

Session E2 Convection and Buoyancy-Driven Flows III: Thermal Instability 324 - Jerzy M. Floryan, University of Western Ontario

4:45PM E2.00001 Stability transitions and energy pathways in horizontal convection at large Rayleigh numbers[1] , BISHAKHDATTA GAYEN, ROSS W. GRIFFITHS, GRAHAM O. HUGHES, Australian National University — We report three-dimensional convective circulation forced by a temperature gradient along the surface of a rectangular channel, using direct and large eddy simulations over a wide range of Rayleigh numbers, $Ra \sim 10^8 - 10^{15}$. The solutions are allowed to reach thermal equilibrium in which there is no net heat input. A sequence of several stability transitions lead to a change from laminar to fully-developed turbulent flow. At the smallest Ra convection is maintained by a balance of viscous and buoyancy forces inside the thermal boundary layer, whereas at the largest Ra inertia dominates over viscous stresses. This results in an enhancement of the overall heat transfer at $Ra \geq 10^{10}$, while both dynamical balances give $Nu \sim Ra^{1/5}$. Our main focus is to analyze the mechanical energy budget. Below the transition the small scales of motion are driven predominately by thermal convection, whereas at $Ra > 10^{13}$ shear plays a dominant role in sustaining the small-scale turbulence.

[1]Numerical computations were conducted using the Australian National Computational Infrastructure, ANU. This work was supported by Australian Research Council grants DP1094542 and DP120102744.

4:58PM E2.00002 Instabilities of Natural Convection in a Periodically Heated Layer , M.Z. HOSSAIN, JERZY M. FLORYAN, University of Western Ontario — Natural convection in a horizontal layer subject to a spatially periodic heating along the lower wall has been investigated. The heating produces sinusoidal temperature variations characterized by the wave number α and the Rayleigh number $\mathrm{Ra_p}$. The primary response has the form of stationary rolls with axis orthogonal to the heating wave vector. For large α convection is limited to a thin layer adjacent to the lower wall with a uniform conduction above it. Linear stability was used to determine conditions leading to a secondary convection. Two mechanisms of instability have been identified. For α =0(1), the parametric resonance dominates and leads to the pattern of instability that is locked-in with the pattern of the heating according to the relation $\delta_{\mathrm{cr}} = \alpha/2$, where δ_{cr} denotes the component of the critical disturbance wave vector parallel to the heating wave vector. The second mechanism, Rayleigh-Bénard (RB) mechanism, dominates for large α. Competition between these mechanisms gives rise to non-commensurable states and appearance of soliton lattices, to the formation of distorted transverse rolls, and to the appearance of the wave vector component in the direction perpendicular to the forcing direction.

5:11PM E2.00003 On the transition to chaos of natural convection between two infinite differentially heated vertical plates[1] , ZHENLAN GAO, LIMSI-CNRS Universite Pierre et Marie Curie, BERENGERE PODVIN, LIMSI-CNRS, ANNE SERGENT, LIMSI-CNRS Universite Pierre et Marie Curie, SHIHE XIN, CETHIL INSA DE LYON, PATRICK LE QUERE, LIMSI-CNRS, LAURETTE TUCKERMAN, PMMH ESPCI — Natural convection of air between two infinite vertical differentially heated plates is studied analytically in two dimensions (2D) and numerically in two and three dimensions (3D), for Rayleigh numbers Ra up to three times the critical value Ra_c. The first instability is a supercritical circle pitchfork bifurcation leading to steady 2D corotating rolls. A Ginzburg-Landau equation is derived analytically for the flow around this first bifurcation and compared with results from direct numerical simulation (DNS). In 2D, DNS shows that the rolls become unstable via a Hopf bifurcation. As Ra is further increased, the flow becomes quasi-periodic, then temporally chaotic for a limited range of Rayleigh numbers, beyond which the flow returns to a steady state through a spatial modulation instability. In 3D, the rolls instead undergo another pitchfork bifurcation to 3D structures, which consist of transverse rolls connected by counter-rotating vorticity braids. The flow then becomes time-dependent through a Hopf bifurcation, as exchanges of energy occur between the rolls and the braids. Chaotic behavior subsequently occurs through two competing mechanisms: a sequence of period-doubling bifurcations leading to intermittency or else a spatial pattern modulation.

[1]Some of the computations were carried out at CNRS-IDRIS Project DARI0326.

5:24PM E2.00004 Spanwise plumes in wakes behind heated cylinder , AJITH KUMAR S, Indian Institute of Technology Madras, Chennai, ANIL LAL S, College of Engineering Trivandrum, SAMEEN A, Indian Institute of Technology Madras, Chennai — 3D wake transition in flow past cylinder is interesting theoretically and industrially. A three dimensional Finite volume computation has been performed on an incompressible flow past heated cylinder to understand the wake behavior behind the cylinder, under the Boussinesq assumption. We study the heat transfer characteristics and the coherent structures behind the cylinder at different Prandtl numbers. In forced convection, the 3D transition occurs above Reynolds number, Re=180-190 (Re is based on the cylinder diameter). However, the present 3D computational analyses show that in mixed convection, the so called "mode-E" instability (3D transition of wake behind the cylinder caused by the heating of the cylinder) happens at a much lower Reynolds number. The co-existence of mushroom like coherent structures called the plumes along with the shed vortices is observed for a range of heating conditions. These plumes originates from the core of the upper vortex rows at a definite span wise wavelengths. The dependence of Prandtl number on the span wise wavenumber of these plumes is also analyzed.

5:37PM E2.00005 Plate-like convection in fluids with temperature-dependent viscosity[1] , ANA M. MANCHO, ICMAT, CSIC, JEZABEL CURBELO, ICMAT, UAM — The study of instabilities in fluids in which viscosity experiences a transition at a certain temperature range is of great interest for the understanding of planetary interiors, since this phenomena models the melting and solidification of a magma ocean and thus is suitable for representing a lithosphere over a convecting mantle. To this end, we study a 2D convection problem in which viscosity depends on temperature by abruptly changing its value by a factor 400 within a narrow temperature gap at which magma melts. We perform a study which combines bifurcation analysis and time dependent simulations. Solutions such as limit cycles are found that are fundamentally related to the presence of the O(2) symmetry. Sporadically during these cycles, through abrupt bursts, spontaneous plate-like behaviors that rapidly evolve towards a stagnant lid regime emerge. The plate-like evolution alternates motions towards either right or left, introducing temporary asymmetries on the convecting styles. Further time dependent regimes are described for different transition laws which are greatly influenced by the presence of the symmetry.

[1]We thank CESGA for computing facilities. This research is supported by the Spanish Ministry of Science under grant MTM2011-26696 and MINECO: ICMAT Severo Ochoa project SEV-2011-0087.

Sunday, November 24, 2013 4:45PM - 6:03PM –

Session E3 Multiphase Flows III 325 - Deniz Tolga Akcabay, University of Michigan

4:45PM E3.00001 Computation of Cavitating Flow in a Francis Hydroturbine , DANIEL LEONARD, Pennsylvania State University, JAY LINDAU, Applied Research Lab/Penn State — In an effort to improve cavitation characteristics at off-design conditions, a steady, periodic, multiphase, RANS CFD study of an actual Francis hydroturbine was conducted and compared to experimental results. It is well-known that operating hydroturbines at off-design conditions usually results in the formation of large-scale vaporous cavities. These cavities, and their subsequent collapse, reduce efficiency and cause damage and wear to surfaces. The conventional hydro community has expressed interest in increasing their turbine's operating ranges, improving their efficiencies, and reducing damage and wear to critical turbine components. In this work, mixing planes were used to couple rotating and stationary stages of the turbine which have non-multiple periodicity, and provide a coupled solution for the stay vanes, wicket gates, runner blades, and draft tube. The mixture approach is used to simulate the multiphase flow dynamics, and cavitation models were employed to govern the mass transfer between liquid and gas phases. The solution is compared with experimental results across a range of cavitation numbers which display all the major cavitation features in the machine. Unsteady computations are necessary to capture inherently unsteady cavitation phenomena, such as the precessing vortex rope, and the shedding of bubbles from the wicket gates and their subsequent impingement upon the leading edge of the runner blades. To display these features, preliminary unsteady simulations of the full machine are also presented.

4:58PM E3.00002 Understanding and Toward Controlling the Hydroelastic Response and Stability of Hydrofoils in Cavitating Flows[1] , DENIZ TOLGA AKCABAY, YIN LU YOUNG, University of Michigan — This study examines the hydroelastic behavior of hydrofoils in cavitating flows. The hydrofoil was modeled as a 2-D structure which can undergo spanwise bending and twisting deformations and the flow is modeled through the unsteady RANS equations. The results show that: 1) amplitude of the dynamic load fluctuations on the hydrofoil can exceed their mean values as the highest load fluctuations and vibrations occur when maximum cavity length is close to foil trailing edge, and the fluctuations decrease when in stable supercavitation; 2) lock-in can lead to dynamic load amplifications and focusing of the frequency at the system resonance frequency, and the system resonance frequency varies with cavitation volume due to changes with fluid added mass; 3) viscous effects, in general, tend to delay static divergence, but unsteady cavity/vortex shedding can lead to flutter and lock-in; 4) the mean hydroelastic loads/deformations are bounded by the stall limit and quasi-steady potential flow estimates for the fully wetted and supercavitating regimes; and 5) transient, viscous fluid-structure interaction models are needed to predict the dynamic response and stability of flexible hydrofoils. Finally, results will be shown to illustrate potential strategies that could be used to minimize cavitation and enhance stability.

[1]This work was supported through the Office of Naval Research (ONR) grant number N00014-11–0833.

5:11PM E3.00003 An acoustically accurate method to simulate turbulent cavitating flows[1] , ASWIN GNANASKANDAN, KRISHNAN MAHESH, University of Minnesota — An acoustically accurate method involving transport of energy equation has been developed to simulate cavitating flows to study sheet to cloud cavitation transition in complex geometries. The algorithm uses a compressible homogeneous equilibrium model and solves transport equation for the vapor's mass fraction along with the compressible Navier-Stokes equation for the mixture. The internal energy equation is demonstrated to discretely outperform the total energy equation. A novel characteristic based filtering method has been developed for multiphase flows and is applied in a predictor-corrector approach, ensuring zero dissipation away from discontinuities. A dynamic Smagorinsky model is used for both Navier Stokes and the scalar transport equation. The algorithm has been validated for a variety of problems. Details of the methodology along with simulation results will be discussed.

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

5:24PM E3.00004 Measurements on a cavitating wedge , HARISH GANESH, SIMO MAKIHARJU, STEVEN CECCIO, University of Michigan, Ann Arbor — Three types of cavitation on a wedge; incipient, transient, and periodic shedding, were chosen to perform extensive flow measurements to establish a reliable experimental dataset for CFD validation. Two time-synchronized high-speed video cameras were used to film the cavitation events from the top and side. A common time base was used to synchronize the cameras with unsteady pressure pulse signature during the cavitation cloud collapse, surface impedance probes implanted on the wedge surface, and flow properties measurement devices. This enabled the possibility to correlate the processes observed in the video to measured flow properties. The whole process was repeated by replacing the high speed video cameras with a time resolved X-ray densitometry system to measure the void fraction distribution synchronously with other flow measurements. Based on all the measurement data, significant physical processes that dominate the cavitation dynamics were identified.

5:37PM E3.00005 Apker Prize Lecture: Using 3D Printing and Stereoscopic Imaging to Measure the Alignment and Rotation of Anisotropic Particles in Turbulence[1] , GUY MARCUS[2], SHIMA PARSA[3], STEFAN KRAMEL, RUI NI, GREG VOTH, Wesleyan University — We have developed a general methodology to experimentally measure the time-resolved Lagrangian orientation and solid body rotation rate of anisotropic particles with arbitrary aspect ratio from standard stereoscopic video image data. We apply these techniques to particles advected in a $R_\lambda \approx 110$ fluid flow, where turbulence is generated by two grids oscillating in phase. We use 3D printing technology to design and fabricate neutrally buoyant rods, crosses (two perpendicular rods), and jacks (three mutually perpendicular rods) with a largest dimension of 7 times the Kolmogorov length scale, which makes them good approximations to tracer particles. We have measured the mean square rotation rate, $\dot{p}_i \dot{p}_i$, of particles spanning the full range of aspect ratios and obtained results that agree with direct numerical simulations. By measuring the full solid-body rotation of jacks, we provide a new, extensible way to directly probe the Lagrangian vorticity of a fluid. We also present direct measurements of the alignment of crosses with the direction of their solid body rotation rate vector—in agreement with direct numerical simulations.

[1]Supported by NSF grant DMR1208990.
[2]Now at Johns Hopkins University
[3]Currently at SEAS, Harvard University

Sunday, November 24, 2013 4:45PM - 6:03PM –
Session E5 CFD III: LES II 327 - Patrick Pisciuneri, University of Pittsburgh

4:45PM E5.00001 Forcing of Wind Turbine Blade Boundary Layer Dynamics by Atmospheric Turbulence with Hybrid URANS-LES[1]

GANESH VIJAYAKUMAR, ADAM LAVELY, BALAJI JAYARAMAN, BRENT CRAVEN, JAMES BRASSEUR, The Pennsylvania State University — We analyze spatio-temporal changes in blade boundary layer structure on a commercial-scale wind turbine blade interacting with a canonical daytime Atmospheric Boundary Layer (ABL). The time scales of the energy-containing ABL eddies are of order multiple rotations of commercial wind turbines and force large temporal fluctuations in integrated loads and bending moments. We study details of blade boundary layer dynamics underlying space-time variations in surface stress by simulating a single blade of the NREL 5MW turbine in a moderately convective ABL produced using LES from a spectral code at high resolution (147M cells). Inflow ABL boundary conditions are extracted for an OpenFOAM ABL simulation with the rotating blade. The blade boundary layer is well resolved with a new hybrid URANS-LES model that blends a 1-equation SFS stress model in the ABL with the k-ω-SST-SAS model near the blade. We perform Hybrid URANS-LES computations of the flow around the blade and compute spatio-temporal fluctuations in surface stresses in response to ABL turbulence eddies. Of particular interest are sources of integrated load transients, load response time scales, and near wake temporal dynamics of vortex shedding in relationship to passage of energy containing atmospheric eddies.

[1]Support: NSF, DOE

4:58PM E5.00002 HPC of Loading Transients on a 5-MW Wind Turbine Rotor by Atmospheric Turbulence Eddies[1]

ADAM LAVELY, GANESH VIJAYAKUMAR, BRENT CRAVEN, BALAJI JAYARAMAN, The Pennsylvania State University, TARAK NANDI, Pennsylvania State University, ERIC PATERSON, Virginia Polytechnic Institute and State University, JAMES BRASSEUR, The Pennsylvania State University — As atmospheric boundary layer (ABL) turbulence eddies sweep through a commercial wind turbine rotor disk, they generate unsteady loadings and bending moments on the blades and shafts. We use blade resolved hybrid-URANS-LES to compute unsteady loadings of a typical daytime moderately convective ABL (MCBL) on the NREL 5 MW wind turbine. The MCBL is generated with LES of exceptional resolution (147M cells) and a low-dissipation spectral algorithm. The ABL LES is used as an initial condition and as inflow boundary conditions for the NREL 5 MW computational domain. This domain is an ABL simulation with 130M cells in an OpenFOAM framework. The atmospheric eddies interact with the blades through a novel hybrid blending of the k-ω-SST-SAS URANS stress model near the blade and a 1-equation SFS LES stress model in the far-field. The time variations in integrated loads, power and bending moments are being correlated with ABL eddy passage. The integrated loads will then be compared to local surface stress transients to determine the source(s) that underlie integrated load, power and moment transients. In this way, we aim to determine the role of atmospheric turbulence on deleterious blade loadings and potential relationships between loadings and local blade boundary layer dynamics.

[1]Support: DOE, NSF

5:11PM E5.00003 Coupling the Actuator Line and Finite Element Methods to Model Fluid Structure Interaction of a Commercial Wind Turbine in the Atmosphere

, JAVIER MOTTA[1], PANKAJ JHA[2], ROBERT CAMPBELL[3], SVEN SCHMITZ[4], JAMES BRASSEUR[5], Pennsylvania State University — Wind turbine blades deform in response to unsteady loadings from atmospheric turbulence, causing changes in local angle-of-attack and blade loadings. This interaction is modeled by a fluid-structure interaction (FSI) solver that combines a finite element (FE) solver with an actuator line method (ALM) model for aerodynamic blade loads and vorticity shedding developed by Jha, et al. (2013). The FSI solver is embedded within an OpenFOAM large-eddy simulation (LES) solver for daytime atmospheric boundary layer (ABL). The flow and structure solvers are tightly coupled to ensure convergence of blade deformation and its impact on the flow field. The structural deformations are computed using a modal summation approach, where the required modal matrix and resonant frequencies are extracted using Abaqus. The ALM and FE algorithms are being optimized to provide a reasonable balance between accuracy of prediction and computation time, particularly due to the sub-iterations required for blade deformation convergence. We also aim to present an analysis of the coupling between blade loading and deformation on the NREL 5MW turbine operating in the ABL. Supported by the DOE.

[1]Graduate Research Assistant, Mechanical Engineering
[2]Graduate Research Assistant, Aerospace Engineering
[3]Assistant Professor, Mechanical Engineering
[4]Associate Professor, Aerospace Engineering
[5]Professor of Mechanical Engineering, Bioengineering, and Mathematics

5:24PM E5.00004 Large Eddy Simulation of a turbulent flow past a wind turbine placed on an undulated wall[1]

, KENNETH CARRASQUILLO, University of Puerto Rico at Mayaguez, STEFANO LEONARDI, Dept. Mechanical Engineering University of Texas at Dallas — With the shortage of fossil fuel and increasing environmental awareness, wind turbines have become the most promising source of renewable energy. A numerical code, solving the Navier-Stokes equations, combined with immersed boundary method and line actuator model has been developed. The Immersed Boundary Method allows to model tower, nacelle and to mimic the topography without the need of body fitted grids. In the actuator line model (ALM), turbine blades are represented by a force distribution on a line which extends from the hub to the tip of the blade. A body force equal and opposite to the lift and drag is imposed in the momentum equation. This force is not imposed in one grid point, instead it is distributed in a volume surrounding the center of the element. Three cases have been considered: one with the turbine blade only, a second set of simulations includes the tower and nacelle on a flat surface and a third simulation presents an undulated wall. Periodic boundary conditions are imposed in the streamwise and spanwise directions. Preliminary results show that the topography on the ground influences the overlying turbulent flow. Roughness affects not only the mean velocity expected at the hub-height, but also fluctuations associated with coherent structures.

[1]This research was supported by the NSF grant # OISE 1243482.

5:37PM E5.00005 Numerical study of the Interaction between Nonsteady Transition and Separation on Oscillating Airfoils[1]

, TARAK NANDI, BALAJI JAYARAMAN, ADAM LAVELY, GANESH VIJAYAKUMAR, The Pennsylvania State University, ERIC PATERSON, Virginia Tech, JAMES BRASSEUR, The Pennsylvania State University — Strong correlation between vertical and horizontal turbulent motions in a daytime atmospheric boundary layer can produce $> 50\%$ variability in local angle of attack(AoA) on commercial wind turbine blade sections. Lee and Gerontakos(JFM 2004)reported an unique experiment where nonsteady transition and boundary layer(BL) separation were estimated on an oscillating airfoil at $Re \sim 10^5$ and reduced frequencies upto 0.2. We use the $k-\omega$ SST URANS model and the $\gamma - Re_\theta$ transition model to explore the predictive capability of these models,and to study the dynamic interactions between transition and separation on an oscillating airfoil with focus on the 3D time-dependent BL characteristics. The calculations are done in OpenFOAM on a wing section of aspect ratio 1 and periodic spanwise boundary conditions. Grid resolution analysis shows that 6M cells are required to resolve the viscous sublayer and capture separation. Fixed AoA cases show good lift comparison but the transition model performs better at higher AoA's when separation-induced transition occurs;fully turbulent URANS mispredicts separation and lift. Prediction of the oscillating cases show differences with experiment in hysteresis loops of the force coefficients. These and related issues will be discussed.

[1]This work is being supported by the DOE

5:50PM E5.00006 A Method for Stable Computations in the Presence of Strong Vortices at Outflow Boundaries[1] , SUCHUAN DONG, Purdue University — We present a robust and accurate outflow boundary condition and an associated numerical algorithm for incompressible flow simulations on severely-truncated unbounded physical domains. This outflow boundary condition allows for the influx of kinetic energy into the domain through the outflow boundaries, and prevents un-controlled growth in the energy of the domain in such situations. The numerical algorithm for the outflow boundary condition is developed on top of a rotational velocity-correction type strategy to de-couple the pressure and velocity computations, and a special construction in the formulation prevents the numerical locking at the outflow boundaries. We show results for several flows with bounded or semi-bounded physical domains, and demonstrate that the presented method produces stable and accurate simulations on even severely truncated computational domains, where strong vortices are present at or exit the outflow boundaries.

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

Sunday, November 24, 2013 4:45PM - 5:50PM –

Session E6 Microfluids: Particles I - Orientation and Self-Assembly 328 - Jerry Shan, Rutgers University

4:45PM E6.00001 Hybrid Opto-electric Manipulation of Macromolecules , STEVE WERELEY, Purdue University — Recently our research group has developed an innovative method for capturing, concentrating, manipulating and sorting populations of micro- and nanometer-scaled entities-particles, cells, macro-molecules, etc. These populations range from individual particles to thousands of particles (Lab-on-a-Chip, 2008; Microfluidics and Nanofluidics, 2008) while the sizes range from microns to nanometers. This novel technique combines features of optical trapping and dielectrophoresis in an innovative, dynamic way using a simple parallel plate electrode configuration. Transparent electrodes comprised of Indium Tin Oxide (ITO) on glass substrates are used to generate an electric field in the fluid while at the same time allowing light into and out of the fluid. Near-IR optical illumination causes subtle localized heating, creating an electric permittivity gradient that in turn drives a microscopic toroidal vortex. The vortex efficiently transports particles to a preferred location, usually the surface of the electrode. Recent advances have extended have allowed us to apply this technique to macromolecules (DNA, proteins) as well as nanoscale particles (quantum dots, nanowires and PSL particles).

4:58PM E6.00002 Customizing mesoscale self-assembly with 3D printing , NICOLAS VANDEWALLE, MARTIN POTY, GEOFFROY LUMAY, GRASP, University of Liege, B-4000 Liege, Belgium — Self-assembly due to capillary forces is a common method for generating 2D mesoscale structures from identical floating particles at the liquid-air interface. Designing building blocks to obtain a desired mesoscopic structure is still a challenge. We show herein that it is possible to shape the particles with a low cost 3D printer, for composing specific mesoscopic structures. Since capillary interactions can be downscaled, our method, for producing capillary multipoles, opens new ways to low cost microfabrication.

5:11PM E6.00003 Electro-orientation of Carbon Nanotubes in Polymer Suspensions , RICHARD CASTELLANO, GABRIEL GIRALDO, CEVAT AKIN, JERRY SHAN, Rutgers University — Carbon nanotube (CNT) membranes have been of recent interest due to experiments and simulations that have found flow rates through nanotubes to be 2 to 3 orders of magnitude faster than predicted by viscous-flow theory. As such, they offer promise as highly permeable membranes for a variety of filtration and separation processes. However, current CNT-membrane fabrication techniques utilize CVD growth of CNTs which is costly and difficult to scale up. Electro-orientation of post-growth CNTs in polymer suspension is a possible cost-effective and scalable approach to producing aligned CNT membranes and composites. An electric field (E-field) applied to a prolate particle induces a dipole in the direction of the particle's major axis which causes the particle to rotate into alignment with the E-field. The alignment rate of CNTs in various polymer suspensions is experimentally studied here as a function of the applied E-field strength and frequency. When dealing with CNTs, thermal energy is a significant component of the particle motion, causing misalignment from the applied E-field. In order to quantify the significance of Brownian motion, we measure the probability distribution of alignment angles for CNTs at various field strengths. We compare our experimental results with theoretical predictions and discuss the implications for producing membranes of aligned CNTs by electro-orientation.

5:24PM E6.00004 A hexatic-to-disorder transition in colloidal crystals near electrodes: Rapid annealing of polycrystalline domains , CARI DUTCHER, TAYLOR WOEHL, NICHOLAS TALKEN, WILLIAM RISTENPART, Dept. Chemical Engineering & Materials Science, University of California, Davis — Colloids are known to form planar, hexagonal closed packed (HCP) crystals near electrodes in response to electrohydrodynamic (EHD) flow. Previous work has established that the EHD velocity increases as the applied AC frequency decreases. Here we report the existence of an order-to-disorder transition at sufficiently low frequencies, despite the increase in the attractive EHD driving force. At large frequencies ($\sim$500 Hz), spherical micron-scale particles form HCP crystals; as the frequency is decreased below $\sim$250 Hz, however, the crystalline structure transitions to randomly closed packed (RCP). The transition is reversible and second order with respect to frequency, and independent measurements of the EHD aggregation rate confirm that the EHD driving force is indeed higher at the lower frequencies. We present evidence that the transition is instead caused by an increased particle diffusivity due to increased particle height over the electrode at lower frequencies, and we demonstrate that the HCP-RCP transition facilitates rapid annealing of polycrystalline domains.

5:37PM E6.00005 Electrokinetically driven reversible self-assembly of colloidal particle bands near the wall[1] , NECMETTIN CEVHERI, MINAMI YODA, Georgia Institute of Technology — Recent studies in microchannels have shown that the near-wall dynamics of neutrally buoyant dielectric colloidal (radii $a < 1$ μm) suspended particles are affected by an electric field of magnitude E applied parallel to the wall. Evanescent-wave particle velocimetry was used to study $a = 245$ nm fluorescent polystyrene particles suspended at volume fractions of $O(10^{-4})$ in combined electroosmotic (EO) and Poiseuille flow of an aqueous electrolyte solution, which is effectively the superposition of simple shear and uniform flows within 0.5 μm of the wall. In "counterflow," where the EO opposes the shear flow through fused-silica microchannels, at a large enough value of E so that flow reversal occurs in the near-wall region, the particles self-assemble into concentrated bright "stripes" along the streamwise direction alternating with dark stripes containing almost no particles with a consistent cross-stream spatial frequency. These stripes are only observed within ~ 1 μm of the wall, and disappear in the absence of an electric field. These observations suggest the existence of a novel electrokinetic instability, and could lead to new methods for controlled self-assembly of particles into anisotropic colloidal crystals.

[1]Supported by NSF and ARO

Sunday, November 24, 2013 4:45PM - 5:50PM –

Session E7 Microfluids: Porous Media 329 - Ho-Young Kim, Seoul National University

4:45PM E7.00001 Regimes of gas transport through macroscopic areas of multi-layer graphene[1] , MICHAEL BOUTILIER, ROHIT KARNIK, CHENGZHEN SUN[2], NICOLAS HADJICONSTANTINOU, MIT — Nanoporous graphene membranes have the potential to surpass the permeance and selectivity limits of current gas separation membranes. Recent experiments and simulations on individual graphene nanopores have demonstrated that molecule-size-selective nanopores can be created and used to separate components of a gas mixture. However, micrometer-scale tears and nanometer-scale intrinsic defects, inherently present in macroscopic areas of graphene, can severely limit the gas separation performance of graphene membranes of practical size. In this study, we measure the inherent permeance of macroscopic, multi-layer graphene membranes to various gases. A model for the transport of gases through these membranes is developed and shown to accurately explain the measured flow rates. The results quantify the separate contributions of tears and intrinsic defects to the inherent permeance of macroscopic areas of multi-layer graphene. The model is then extended to graphene membranes with engineered selective nanopores to optimize design parameters for defect-tolerant gas separation membranes.

[1]This work was supported in part by the MIT Energy Initiative and in part by a NSERC PGS-D3 Fellowship
[2]Also at Xi'an Jiaotong University, Xi'an, China

4:58PM E7.00002 Molecule permeation and gas separation by nanoporous graphene membranes[1] , NICOLAS HADJICONSTANTINOU, CHENGZHEN SUN[2], MICHAEL BOUTILIER, ROHIT KARNIK, MIT — Molecular simulations and experiments suggest that by introducing nanopores of appropriate size, nanoporous graphene membranes can exhibit permeability and selectivity exceeding those of existing state-of-the-art membranes by several orders of magnitude. To better understand how gases permeate through these membranes, we conducted molecular dynamics simulations of gas permeation through different nanopores for four different gases, namely helium, hydrogen, nitrogen and methane. Our results show that in addition to the direct flux, defined as the contribution from molecules crossing directly from the gas-phase on one side of the graphene to the other, in some gases, significant contribution to the flux across the membrane comes from a surface mechanism, in which the molecules cross after being adsorbed onto the graphene surface. Our results quantify the relative contribution of the bulk and surface mechanisms and show that the direct flux can be described reasonably accurately using kinetic gas theory, provided the latter is appropriately modified to account for finite-molecule-size effects by assuming steric interactions between rigid pores and hard-sphere gas molecules of known kinetic diameters.

[1]This work was supported in part by the MIT Energy Initiative and the China Scholarship Council
[2]Also at Xi'an Jiatong University, Xi'an, China

5:11PM E7.00003 Dehydration induced phase transitions in a microfluidic droplet array for the separation of biomolecules , CHRIS NELSON, SHELLEY ANNA, Carnegie Mellon University — Droplet-based strategies for fluid manipulation have seen significant application in microfluidics due to their ability to compartmentalize solutions and facilitate highly parallelized reactions. Functioning as micro-scale reaction vessels, droplets have been used to study protein crystallization, enzyme kinetics, and to encapsulate whole cells. Recently, the mass transport out of droplets has been used to concentrate solutions and induce phase transitions. Here, we show that droplets trapped in a microfluidic array will spontaneously dehydrate over the course of several hours. By loading these devices with an initially dilute aqueous polymer solution, we use this slow dehydration to observe phase transitions and the evolution of droplet morphology in hundreds of droplets simultaneously. As an example, we trap and dehydrate droplets of a model aqueous two-phase system consisting of polyethylene glycol and dextran. Initially the drops are homogenous, then after some time the polymer concentration reaches a critical point and two phases form. As water continues to leave the system, the drops transition from a microemulsion of DEX in PEG to a core-shell configuration. Eventually, changes in interfacial tension, driven by dehydration, cause the DEX core to completely de-wet from the PEG shell. Since aqueous two phase systems are able to selectively separate a variety of biomolecules, this core shedding behavior has the potential to provide selective, on-chip separation and concentration.

5:24PM E7.00004 Buckling of a colloid-armored bubble[1] , NICOLAS TACCOEN, LadHyX and Department of Mechanics, Ecole Polytechnique, CNRS, Palaiseau 91128, France, DENIZ Z. GUNES, Nestle Research Center, Food Science and Technology Department, Vers-Chez-Les-Blanc, CH-1000 Lausanne 26, Switzerland, CHARLES N. BAROUD, LadHyX and Department of Mechanics, Ecole Polytechnique, CNRS, Palaiseau 91128, France — We investigate the dissolution of a single air-in-water bubble whose surface is coated with solid particles, as an elementary model of an aging particle-stabilized foam. A microfluidic setup is used to produce a single bubble on demand, force the adsorption of particles to its interface, and hold it stationnary for long-term observation. When the gas dissolves in the surrounding liquid, the particles on the interface eventually jam, thus forming a rigid shell that encloses the bubble. As the temperature and pressure conditons are varied, this armor can either arrest the dissolution of the gas or it can buckle, which leads to the complete disapearance of the bubble. We experimentally demonstrate the existence of a threshold pressure above which the shell is not resistant enough to stabilize the bubble. This is modeled by comparing the mechanical resistance of the hollow shell with the compressive stress due to the dissolution in the liquid, which is controlled through the thermodynamic parameters. These experiments yield the first quantitative measurements of the mechanical resistance of a colloidal shell against ripening. It opens the possibility to study the behavior of more complex armors, by varying the size distribution, the shape and the chemistry of the particles.

[1]Project partly funded by the ERC/FP7 grant No. 278248 "Multicell."

5:37PM E7.00005 Does the hourglass shape of aquaporins optimize water permeability?[1] , SIMON GRAVELLE, LAURENT JOLY, FRANÇOIS DETCHEVERRY, CHRISTOPHE YBERT, CECILE COTTIN-BIZONNE, LYDERIC BOCQUET, Institut Lumiere Matiere, LIQUIDE ET INTERFACES TEAM — The ubiquitous aquaporin channels are able to conduct water across cell membranes, combining the seemingly antagonist functions of a very high selectivity with a remarkable permeability. While molecular details are obvious keys to perform these tasks, the overall efficiency of transport in such nanopores is also strongly limited by viscous dissipation arising at the connection between the nanoconstriction and the nearby bulk reservoirs. In this contribution, we focus on these so-called entrance effects and specifically examine whether the characteristic hourglass shape of aquaporins may arise from a geometrical optimum for such hydrodynamic dissipation. Using a combination of finite element calculations and analytical modeling, we show that conical entrances with suitable opening angle can indeed provide a large increase of the overall channel permeability. Moreover, the optimal opening angles that maximize the permeability are found to compare well with the angles measured in a large variety of aquaporins. This suggests that the hourglass shape of aquaporins could be the result of a natural selection process toward optimal hydrodynamic transport. Finally, in a biomimetic perspective, these results provide guidelines to design artificial nanopores with optimal performances.

[1]This research was supported by the ERC program, project Micromegas.

Sunday, November 24, 2013 4:45PM - 5:50PM –

Session E8 Particle-Laden Flows III: Particle-Turbulence Interaction 330 - David Richter, University of Notre Dame

4:45PM E8.00001 Feedback effect on the large-scale fluid motion in wall-bounded gas-solid disperse flow , YOICHI MITO, Kitami Institute of Technology — Influence of the forces, exerted by dispersed particles, in a channel, in which gas is flowing turbulently, is examined using a direct numerical simulation to calculate the gas velocities seen by the particles and a point force method to calculate the forces exerted by the particles on the gas. Influence of gravity and inter-particle collisions is ignored. Distributions of the mean streamwise body forces, exerted on the fluid by the turbulence and by the particles, are calculated to show the mean large-scale motions of the fluid phase and of the disperse phase. The fluid turbulence forces decrease with increasing volume fraction to accommodate the inter-phase body forces. Thus the large-scale fluid motions, which make a major contribution to the fluid turbulence, are damped. The turbophoretic velocities, which represent the mean drifts, show that mean contribution of each particle to the mean large-scale motion of the disperse phase decreases with increasing volume fraction. This is caused by the decreases in the fluid turbulence and the turbulent transport, with increasing volume fraction.

4:58PM E8.00002 Effects of small particles on coherent structures in particle-laden near-wall turbulence , JUNGHOON LEE, CHANGHOON LEE, Department of Computational Science and Engineering, Yonsei University, Seoul, Korea — In near-wall turbulence, particles interact effectively with coherent structures, such as the quasi-streamwise vortices near the wall. The quasi-streamwise vortices play a significant role in turbulence production and regeneration. In this study, we investigate the modification of the quasi-streamwise vortices due to the presence of particles using direct numerical simulation of turbulent channel flow. The particles considered are smaller than the Kolmogorov length scale and the particle Reynolds numbers are small. Therefore, a point-force approach was used in imposing the particle reaction force on the fluid. Since particles are assumed to be heavier than the fluid, the particle equation of motion was established considering only Stokes drag. In this study, the particle Stokes numbers based on wall units range from 0.5 to 25. It is shown that particles with the lowest Stokes number augment turbulence while particles with higher Stokes numbers attenuate it. The lowest-Stokes-number particles are found to enhance the low- and high-speed streaks around the quasi-streamwise vortices, affecting vortex regeneration cycle. Consequently, the frequency of the quasi-streamwise vortices is increased. However, particles with higher Stokes numbers directly damp the quasi-streamwise vortices.

5:11PM E8.00003 Laminar-turbulent transition of channel flows: the effect of neutrally buoyant finite-size particles[1] , MICHELINE ABBAS, VINCENT LOISEL, OLIVIER MASBERNAT, University of Toulouse - Laboratoire de Genie Chimique, ERIC CLIMENT, University of Toulouse - Institut de Mecanique des Fluides de Toulouse — Numerical simulations were performed on channel flows laden with resolved finite-size neutrally buoyant particles at moderate volumetric concentration. In the case of fluctuating flows close to laminar-turbulent transition, the particle volume fraction is homogeneously distributed in the channel except an accumulation layer in the near-wall region (particle migration is driven by inertia). Particles increase the level of perturbations close to the wall leading to significant enhancement of both the velocity fluctuations and the wall friction coefficient. Additionally, particles break down the large-scale flow structures into smaller, more numerous and sustained eddies. When the flow Reynolds number is decreased, flow relaminarization occurs at critical Reynolds number Re_{cS} (based on the effective suspension viscosity) significantly below the critical Reynolds number Re_c of single-phase flow transition. In the case of laminar flows, the suspension segregates into pure fluid and particle laden wall layers due to cross-stream migration. An instability is observed characterized by the formation of dune-like patterns at the separation between pure fluid and concentrated suspension. Increasing the Reynolds number yields transition to turbulence at a threshold above Re_{cS}.

[1]This work was granted access to the HPC resources of CALMIP and GENCI under the allocations 2012-P1002 and x20132a6942 respectively.

5:24PM E8.00004 Anisotropy of inertial-particle clustering in homogeneous turbulent shear flow[1] , PARVEZ SUKHESWALLA, LANCE COLLINS, Cornell University — We study the clustering of inertial particles dispersed in homogeneous turbulent shear flow (HTSF), with a view towards characterizing the effects of flow-anisotropy on clustering as a function of Stokes numbers, separation distance, and time. Recent experiments [Nicolai et al., *Phys. Fluids* (in review)] have shown preferential orientation of clusters along the plane of maximum mean-strain, for separations larger than the Kolmogorov scale (η). High-resolution ($2048 \times 1024 \times 1024$ grid) direct numerical simulations at similar flow conditions are performed using a hybrid Pseudospectral-WENO scheme, that allows well-resolved, long-time simulations of HTSF at high Reynolds numbers. Inertial particles at different Stokes numbers are tracked, and their angular distribution functions (ADFs) are analyzed. Consistent with Nicolai et al., we observe the particle concentrations are maximal along the extensional axis of the strain component of the imposed uniform mean shear. We quantify the anisotropy by the harmonic decomposition of the ADFs. The first harmonic is found to peak between 5 and 10η for all particle classes. The results pave the way for future studies of the role anisotropy plays in aerosol processes such as collision and gravitational settling.

[1]This work was supported by NSF grant CBET-0967349, and the simulations were supported by XSEDE under NSF grant OCI-1053575.

5:37PM E8.00005 Preferential Concentration Driven Instability of Sheared Gas-Solid Suspensions , MOHAMED KASBAOUI, Sibley School of Mechanical and Aerospace Engineering, Cornell University, DONALD KOCH, School of Chemical and Biomolecular Engineering, Cornell University, GANESH SUBRAMANIAN, JNCASR, Bangalore, India, OLIVIER DESJARDINS, Sibley School of Mechanical and Aerospace Engineering, Cornell University — Through a linear stability analysis of a gas-solid suspension of particles with low Stokes number and moderate mass loading, we demonstrate that the modulation of the gravitational force exerted on the suspension due to preferential concentration of particles in regions of low vorticity can destabilize a homogeneous linear shear flow of a gas-solid suspension. Since the fastest growing modes are found to be those with wavelengths small compared with the characteristic length scale (U/Γ) where U is the settling velocity and Γ is the shear rate, we apply an asymptotic multiple scale analysis using the WKB method. This analysis reveals that the instability comes from the coupling of a particle number density mode driven by preferential concentration in regions where the velocity disturbance reduces the base state vorticity and a momentum mode driven by the particle number density variations. The growth of the amplitude of particle concentration and fluid velocity disturbances is characterized as a function of the wave number and Reynolds number using both the asymptotic theory and a numerical solution of the linearized equations.

Sunday, November 24, 2013 4:45PM - 5:50PM –

Session E9 Biofluids: General III - Pumping Phenomena 333 - Anne Staples, Virginia Polytechnic Institute and State University

4:45PM E9.00001 A bioinspired pumping model for flow in a microtube with rhythmic wall contractions[1] , YASSER ABOELKASSEM, Yale University, ANNE STAPLES, Virginia Tech — Inspired by respiratory systems in insects, in particular the rhythmic wall contractions found in insect tracheal tubes, we propose a bioinspired pumping model that can work particularly well in the low Reynolds number flow regime. Incompressible, viscous flow transport in a fluid-filled axisymmetric, inelastic tube with rhythmic wall contractions is modeled using lubrication theory. The wall motions are prescribed via a tube profile with two contraction sites that can move with a time lag with respect to each other. The analytical model is validated using the method of fundamental solutions based on the Stokeslets meshfree computational method. The velocity field, pressure, and time averaged net flow rate induced in a complete contraction cycle are calculated. The results demonstrate that an inelastic tube with at least two contraction regions (collapse sites) can produce unidirectional flow and working as pumping mechanism. We believe that the physical mechanism underlying the pumping observed in this model relies on the cyclical, temporally asymmetric resistance to upstream and downstream flow that the localized contraction sites exert on one another.

[1]This material is based upon work supported by the National Science Foundation under Grant No. EFRI-BSBA 0938047

4:58PM E9.00002 Electro-dynamic suction pumping at small scales , AUSTIN BAIRD, LAURA MILLER, UNC: Chapel Hill — Dynamic suction pumping is characterized by a bidirectional elastic wave and a non-linear frequency flow relationship. This pumping mechanism has been proposed as the driving mechanism for the vertebrate embryonic heart at the tubular stage. In this study, we consider the tubular, valveless heart of a chordate, the Ascidian *Clavelina picta*. These hearts operate at a Womersley number of about 0.3. We investigate traditional dynamic suction pumping on these small scales and show computationally and experimentally that significant flow isn't achieved. We propose a different pumping mechanism that couples traveling waves of depolarization to the contraction of the boundary. Active contractile waves replace passive elastic waves, but the resulting kinematics are similar to dynamic suction pumping. This pumping mechanism can be computationally shown to drive fluid flow at the low Womersley numbers found in Ascidian hearts.

5:11PM E9.00003 Urinal Dynamics , RANDY HURD, KIP HACKING, BENJAMIN HAYMORE, TADD TRUSCOTT, Brigham Young University, SPLASH LAB TEAM — In response to harsh and repeated criticisms from our mothers and several failed relationships with women, we present the splash dynamics of a simulated human male urine stream impacting rigid and free surfaces. Our study aims to reduce undesired splashing that may result from lavatory usage. Experiments are performed at a pressure and flow rate that would be expected from healthy male subjects.[1] For a rigid surface, the effects of stream breakup and surface impact angle on lateral and vertical droplet ejection distances are measured using high-speed photography and image processing. For free surface impact, the effects of velocity and fluid depth on droplet ejection distances are measured. Guided by our results, techniques for splash reduction are proposed.

[1]Lapides, J., Fundamentals of Urology, W.B. Saunders, Philadelphia, 1976.

5:24PM E9.00004 The Hydrodynamics of Urination: to drip or jet , JONATHAN PHAM, PATRICIA YANG, JEROME CHOO, DAVID HU, Georgia Institute of Technology — The release of waste products is fundamental to all life. How are fluids released from the body quickly and efficiently? In a combined experimental and theoretical investigation, we elucidate the hydrodynamics of urination across five orders of magnitude in animal mass. Using high-speed videography and flow-rate measurement at the Atlanta Zoo, we report discrete regimes for urination style. We observe dripping by small mammals such as rats and jetting by large mammals such as elephants. We discover urination duration is independent of animal size among animals that use jetting. We rationalize urination styles, along with the constant-time scaling, by consideration of the relative magnitudes of the driving forces, gravity and bladder pressure, and the corresponding viscous losses within the urethra. This study may give insight into why certain animals are more prone to diseases of the urinary tract, and how the urinary system evolved under the laws of fluid mechanics.

5:37PM E9.00005 The role of amniotic fluid in force transfer during human birth , ALEXA BAUMER, ANDREA LEHN, MEGAN LEFTWICH, The George Washington University — This study seeks to understand the fundamental fluid dynamic processes involved in human birth. We begin by examining the importance of amniotic fluid. This is done using two experimental techniques that approximate the laboring human uterus to different degrees of anatomical correctness. The first, in which a latex uterus is filled with fluid and a solid fetus is extracted, investigates the importance of both amniotic fluid properties and fetal position in the force required to remove a fetus. The second experiment simplifies the geometry of birth even more. In this case, a solid cylindrical rod is pulled through a highly flexible outer tube. The force to pull the inner cylinder as a function of the gap fluid properties is measured. By carefully controlling the fluid properties of the experiment, the study will provide further insight into the roles of amniotic fluid in human birth.

Sunday, November 24, 2013 4:45PM - 5:50PM –

Session E10 Jets II 334 - Mark Glauser, Syracuse University

4:45PM E10.00001 Reduced-order modeling of wavepackets in controlled jets with corrugated profiles , ANIRUDDHA SINHA, California Institute of Technology, ALI UZUN, Florida State University, TIM COLONIUS, California Institute of Technology — Reduced noise from high-speed turbulent jets has been achieved through passive and active devices distributed uniformly around the nozzle lip. These devices include chevrons as well as a spinning valve fluidic injection actuator that can deliver steady or harmonic excitation. On a time-averaged basis, all these actuators introduce corrugations in the mean shear layer, and our analysis shows that such corrugations give rise to new instability mechanisms in addition to the usual Kelvin-Helmholtz modes. We study these controlled jets using parallel-flow linear stability analysis, as well as the theory of linear parabolized stability equations (PSE) that accounts for mild streamwise variations of the mean flow. For unforced (round) subsonic and supersonic jets, PSE models have previously been shown to give accurate predictions for the large-scale coherent structures (wavepackets) educed from experimental data. We examine similar data for forced jets, as well as jets issuing from serrated nozzles, and seek to explain the observed modifications in wavepacket dynamics and the reduced noise radiation as changes in the instability characteristics of the corrugated mean flows.

4:58PM E10.00002 Towards High Speed Jet Noise Reduction Using Time-Resolved PIV[1] , ZACHARY BERGER, Syracuse University, MATTHEW BERRY, Syracusye University, PATRICK SHEA, Syracuse University, BARRY KIEL[2], Air Force Research Laboratory, NAIBO JIANG, Spectral Energies, LLC., BERND NOACK, Institute PPRIME/CNRS, SIVARAM GOGINENI, Spectral Energies, LLC., MARK GLAUSER, Syracuse University — In this investigation, the flow field of a Mach 0.6 turbulent, compressible jet is studied using time-resolved particle image velocimetry (TRPIV). The hydrodynamics and acoustics are simultaneously sampled using pressure sensors in the near-field and microphones in the far-field, respectively. Two-component velocity measurements are taken in the streamwise plane of the jet, just before the collapse of the potential core. Several planes are obtained off of the jet's centerline, providing information across the entire nozzle and beyond the expanding shear layer. These measurements will provide a three-dimensional view of the flow field in the spanwise direction of the jet. Low-dimensional modeling tools are implemented to extract the energetic modes in the flow. In addition, correlations between the near-field velocity and the far-field acoustics are computed using similar techniques. These results will assist in identifying the structures and events in the near-field responsible for the far-field noise. The goal is to use the time-evolution of the flow field to identify these events both spatially as well as temporally. Ultimately, active flow control schemes will then be developed based on these findings.

[1]Spectral Energies, LLC. and AFRL
[2]Wright-Patterson Air Force Base

5:11PM E10.00003 Analysis and comparison of non-axisymmetric and circular nozzle configurations of a high speed jet[1] , MATTHEW BERRY, Syracuse University, BARRY KIEL, AFRL, ZACHARY BERGER, Syracuse University, NAIBO JIANG, SIVARAM GOGINENI, Spectral Energies LLC, MARK GLAUSER, Syracuse University — The main focus of this experiment is on the analysis and comparison of different nozzle configurations of a high speed jet. We used 3 different non-axisymmetric nozzle designs placed in 7 different orientations as well as our standard 2 inch circular nozzle. The flow field was investigated at Mach 0.6 using two-component time-resolved PIV simultaneously sampled with far-field acoustic measurements. The velocity was examined in the streamwise direction of the r-z plane using 10 kHz TRPIV with a window size of about 5 inches. An array of 12 G.R.A.S. microphones were placed about 150 inches from the jet nozzle. Low-dimensional modeling was performed on the velocity data to extract and compare the structures associate with the different nozzle designs. The overall sound pressure level at several polar angles with respect to the jet axis was also compared.

[1]Spectral Energies and AFRL

5:24PM E10.00004 The influence of the density ratio on the linear frequency response of low-density jets[1] , WILFRIED COENEN, ALEJANDRO SEVILLA, Área de Mecánica de Fluidos, Dpto. de Ingeniería Térmica y de Fluidos, Universidad Carlos III de Madrid, Spain, LUTZ LESSHAFFT, Laboratoire d'Hydrodynamique (LadHyX), École Polytechnique - CNRS, France — Low-density jets support global self-sustained oscillations when the jet-to-ambient density ratio is sufficiently small, a phenomenon that has been linked to the presence of a region of local absolute instability in the underlying parallel base flow. However, the use of local stability analysis requires introducing ad-hoc criteria to match the experimental observations (see Coenen & Sevilla, J. Fluid Mech. 713, 2012, and references therein). In this work we therefore use a global approach, where the wavepacket structures are temporal eigenmodes of the linearized equations of motion in a 2D domain. The resulting eigenvalue spectra show that, when the density ratio is decreased, a discrete eigenmode becomes increasingly dominant, eventually reaching a positive growth rate for a certain critical density ratio. For the particular case of a He/air jet, this critical density ratio, as well as the corresponding oscillation frequency, is in good quantitative agreement with the experiments of Hallberg & Strykowski (J. Fluid Mech. 569, 2006). The influence of the density ratio on the linear frequency response of the jet under globally stable conditions is also investigated.

[1]Supported by Spanish MINECO under project DPI 2011-28356-C03-02.

5:37PM E10.00005 Structure of backward facing step flow in low Reynolds number controlled by synthetic jet array with different injection velocities , SANEYUKI TAKANO, Tokyo University of Science — This study presents detailed structure of separated flow downstream of a backward facing step affected by a non-uniform periodic disturbance along spanwise direction induced by synthetic jet array. The Reynolds number based on the step height ranged from 300 to 900. The frequency of the synthetic jet actuation was selected within the acceptance frequency range of separating shear layer. The periodic disturbance generates periodic transverse vortices whose size and shape change corresponding to the strength of the disturbance. The effect of different injection velocities in the synthetic jet array from those of adjacent jets on the transverse vortex structure and resulting reattachment process is discussed based on the wall shear stress measured by the Micro Flow Sensor (MFS) and flow visualization. Near wall behavior of the transverse vortex above the MFS was related to the sensor output. The results show that non-uniform injection velocity manipulated in the jet array induces difference in the distorted vortex structure and reattachment process in spanwise direction, which strongly depend on the Reynolds number and injection velocities of the synthetic jets.

Sunday, November 24, 2013 4:45PM - 5:37PM –

Session E11 Bubbles III: Soap, Films and Foams 335 - Kevin Connington, City College of New York

4:45PM E11.00001 How are soap bubbles blown? Fluid dynamics of soap bubble blowing , JOHN DAVIDSON, LORI LAMBERT, ERICA SHERMAN, TIMOTHY WEI, SANGJIN RYU, University of Nebraska-Lincoln — Soap bubbles are a common interfacial fluid dynamics phenomenon having a long history of delighting not only children and artists but also scientists. In contrast to the dynamics of liquid droplets in gas and gas bubbles in liquid, the dynamics of soap bubbles has not been well documented. This is possibly because studying soap bubbles is more challenging due to there existing two gas-liquid interfaces. Having the thin-film interface seems to alter the characteristics of the bubble/drop creation process since the interface has limiting factors such as thickness. Thus, the main objective of this study is to determine how the thin-film interface differentiates soap bubbles from gas bubbles and liquid drops. To investigate the creation process of soap bubbles, we constructed an experimental model consisting of air jet flow and a soap film, which consistently replicates the conditions that a human produces when blowing soap bubbles, and examined the interaction between the jet and the soap film using the high-speed videography and the particle image velocimetry.

4:58PM E11.00002 Coalescence of soap bubbles: petals and fractals , BENG HAU TAN, SILVESTRE ROBERTO GONZALEZ AVILA, CLAUS-DIETER OHL, Nanyang Technological University — The coalescence of thin film bubbles, i.e. soap bubbles, is determined by successive ruptures of the two films approaching each other. Ruptures in isolated thin films have been studied experimentally in detail and their dynamics is well understood theoretically; less so for the coalescence of soap bubbles. In this case, the film rupture occurs in very close proximity to a second film. The interaction between one quickly retracting film with a stationary film leads to complex dynamics. High-speed photography of the events occurring on a microscopic scale is conducted. We find that within the first 100 microseconds radially symmetric fingering and fractal structures are created at the rupture site. The first film retraction may induce the rupture of the second film. Later the retracting soap film causes the entrainment of a ring of secondary bubbles and possibly droplets along its circumference. Some first modelling will be presented, too.

5:11PM E11.00003 Plastic and Elastic Deformations of Foam Bubbles Driven by Oscillatory Compression[1] , KLEBERT FEITOSA, NICHOLAS HAGANS, CHRISTINE O'DEA, Dept. of Physics and Astronomy, James Madison University — Fluidization of two-dimensional (2D) foam is characterized by rearrangement events known as T1-events where clusters of four bubbles switch neighbors. This research focus on rearrangement events of bubbles in a bubble raft subject to periodic compression by an oscillating boundary. The instantaneous position of the bubbles are tracked from images of the bubble raft captured with a high speed camera. We find that T1-events are reversible for small amplitude oscillations (elastic deformations), but irreversible for large amplitude oscillations (plastic deformations). We also find that T1 events are spatially correlated confirming that such rearrangements leads to local fluidization.

[1]Research Corporation

5:24PM E11.00004 Crack Propagation Dynamics and Film Instability in Liquid Foams , SASCHA HILGENFELDT, Mechanical Science and Engineering, University of Illinois at Urbana-Champaign, PETER STEWART, Oxford Centre for Collaborative Applied Mathematics, Mathematical Institute, The University of Oxford, STEPHEN DAVIS, Department of Engineering Sciences and Applied Mathematics, Northwestern University — Quasi-two-dimensional liquid foams (a single layer of foam bubbles between parallel plates) are model systems for the behavior of solid-state materials, including their flow and failure. Upon introduction of pressurized air, the foam layer was shown to yield and fail in two different ways, analogous to ductile and brittle fracture. The microscopic processes of deformation, plasticity, and loss of cohesion on the bubble scale are accessible in detail to experiment and modeling, using elements of fluid dynamics, stability theory, and surface chemistry. A simplified network model of liquid nodes captures both fracture modes and allows for quantitative assessment of microscopic effects. For the brittle crack propagation, which involves breakage of a succession of thin liquid films, we show that viscosity and Marangoni stresses can play significant roles in determining film instability and thus the time scales of foam failure, with important consequences for practical applications such as metal foam manufacture or oil recovery.

Sunday, November 24, 2013 4:45PM - 5:50PM –

Session E12 Vortex Dynamics and Vortex Flows III 336 - Stavros Tavoularis, University of Ottawa

4:45PM E12.00001 Jet Interactions in a Feedback-Free Fluidic Oscillator in the Transition Region , MEHMET TOMAC, Abdullah Gul University, JAMES GREGORY, The Ohio State University — The details of the jet interactions and oscillation mechanism of a feedback-free type fluidic oscillator are studied in this work. Flow rate-frequency measurements indicate the existence of three distinct operating regimes: low flow rate, transition, and high flow rate regions. This study presents results from the transition regime, extracted by using refractive index-matched particle image velocimetry (PIV). A newly-developed sensor configuration for frequency measurements in the refractive index-matched fluid and a phase-averaging method that minimizes jitter will be discussed. Experimental results indicate that the interactions of the two jets create three main vortices in the mixing chamber. One vortex vanishes and forms depending on the oscillation phase and plays a key role in the oscillation mechanism. The other two vortices sustain their existence throughout the oscillation cycle; however, both continuously change their size and strength. The resulting complex flow field with self-sustained oscillations is a result of the combination of many interesting phenomena such as jet interactions and bifurcations, viscous effects, vortex-shear layer interactions, vortex-wall interactions, instabilities, and saddle point creations.

4:58PM E12.00002 Characterization of self-excited fluidic energy harvesters in uniform flows[1] , VAHID AZADEH RANJBAR, CORALIE CLER, NIELL ELVIN, YIANNIS ANDREOPOULOS, The City College of The City University of New York — Energy harvesters consisting of a low aspect-ratio hollow circular cylinder attached to the free end of a cantilevered beam which is partially covered by piezoelectric patches near its clamped end to produce electrical power output have been investigated experimentally and analytically. The unsteady nature of vortex shedding is described by the van der Pol equation, a non-conservative oscillator with non-linear damping, which models the near wake dynamics that is coupled with the harvester's equation of motion. This model helps to describe and predict the vortex induced vibration phenomena such as lock-in range, maximum amplitude of oscillations and extension of structural oscillations far away lock-in range with a better physical insight. Both free vibration and wind tunnel tests were carried out to characterize the harvester. Based on the wind tunnel tests data, there is a remarkable difference in magnitude and frequency of the lift force between stationary and oscillating cylinders subjected to stationary uniform flow. Moreover, maximum electrical power output occurs at a forcing frequency somewhat higher than the structural resonance frequency. These experimental results are in good agreement with the results of the mathematical model.

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

5:11PM E12.00003 Fluid-membrane dynamics of energy harvesting eel immersed in a square cylinder wake , YING ZHENG LIU, YUE LONG YU, School of Mechanical Engineering, Shanghai Jiao Tong University — A comprehensive study was performed of fluid-membrane dynamics of energy harvesting eel placed behind a square cylinder. A low aspect-ratio of PVDF membrane was employed for the experiments in a low-speed wind tunnel; simultaneous measurements of the closed loop voltage and the flapping motion were made by using a digital oscilloscope and a high-speed camera, respectively. The experimental Reynolds number based on the width of the cylinder was Re= 1,000-10,000. Influence of electric resistances on the power output was determined by varying the electric resistances in the closed loop. A state-of-the-art algorithm of image analysis, which was proposed for accurately identifying arbitrarily moving and deforming boundary, was performed on the consecutively recorded membrane, resulting in the digitized data of the flapping membrane. The pattern of the flapping membrane under different Reynolds numbers and electric resistances was compared. Time history of the membrane tip deflection, power spectral determined at different locations along the length of the membrane, spectral feature of the voltage were demonstrated for correlation analysis.

5:24PM E12.00004 Estimating the pressure forcing on a flexible piezoelectric beams exerted by a passing vortex using time-resolved PIV data[1] , OLEG GOUSHCHA, NIELL ELVIN, YIANNIS ANDREOPOULOS, The City College of The City University of New York — A cantilever flexible beam instrumented with a piezoelectric patch and immersed in a flow can be used to harvest fluidic energy. Pressure distribution induced by naturally present vortices in a turbulent fluid flow can force the beam to oscillate producing electrical current. Maximizing the power output of such an electromechanical fluidic system is a challenge. In order to understand the pressure force exerted on the beam in a fluid flow where vortices of different scales are present, an experimental facility was set up to observe the interaction of individual vortices with the beam and record the time-resolved PIV data around the beam. Using the time-resolved PIV data, the pressure Poisson equation is solved by using a Green function's approach to obtain the pressure distribution over the beam. The beam is instrumented at the base with a piezoelectric patch, a strain gage and a force sensor whose output data are compared to the results from the pressure Poisson equation solution. A large negative pressure peak is observed as the vortex core travels over the beam responsible for the net lift force deflecting the beam towards the center of the vortex core.

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

5:37PM E12.00005 Energy Harvesting from an Oscillating Flat Plate in a Uniform Flow , BENJAMIN STROM, Brown University, JENNIFER FRANCK, KENNETH S. BREUER, Brown University, Providence RI — A flat plate of aspect ratio 4.12 was pitched sinusoidally about the center chord in a uniform flow over a range of frequencies, amplitudes and mean angles of attack with the objective of studying the system's energy harvesting capabilities. Energy transfer from the fluid to plate, calculated from the torque and angular position, was found to be positive over a wide range of pitching parameters with an optimal mean angle of attack of approximately 15 degrees. Energy transfer per cycle was found to correlate linearly with a proposed measure of the total circulation in the leading edge vortex. The characteristic length determining the leading edge vortex strength appeared to differ for angles of attack less than and greater than the optimum. Comparisons with unsteady Large Eddy Simulations give insight into the vortex dynamics and mechanisms of energy transfer.

Sunday, November 24, 2013 4:45PM - 5:50PM –

Session E13 Focus Session: Marine Hydrokinetic Energy Conversion III 301 - Luksa Luznik, United States Naval Academy

4:45PM E13.00001 Flow Structures and Energy Capture from an Oscillating Hydrofoil , JENNIFER FRANCK, SARAH FRANK, SHREYAS MANDRE, Brown University — The flow surrounding an oscillating hydrofoil in a uniform freestream is computationally investigated for hydrokinetic energy capture. Simulations are performed on an elliptical hydrofoil using 2D Direct Numerical Simulation (DNS) for low Reynolds number and 3D Large-Eddy Simulations (LES) for high Reynolds number simulations at 80,000. A non-inertial reference frame is utilized for rigid-body motion of the hydrofoil, which is prescribed a sinusoidal motion in pitch and heave. The kinematic parameters are varied and the resulting flow features are correlated with positive or negative energy capture. In an effort to optimize the stroke, variations in the sinusoidal heave and pitch signals are systematically explored and analyzed for future closed-loop control.

4:58PM E13.00002 On the effects of turbine geometry on the far wake dynamics of an axial flow hydrokinetic turbine[1] , FOTIS SOTIROPOULOS, XIAOLEI YANG, St. Anthony Falls Laboratory, Dept. of Civil Engineering, University of Minnesota, SEOKKOO KANG, Dept. of Civil Engineering, Hanyang University, South Korea — In large-eddy simulation (LES) of multi-turbine arrays actuator disk (AD) or actuator line (AL) models are employed to simulate individual turbines. Such parameterizations do not take into account the details of the turbine geometry and, therefore, cannot be expected to accurately resolve the flow in the near wake. We investigate the performance of AD and AL models by comparing their predictions with laboratory measurements and with LES resolving the geometrical details of the turbine. We simulate the flow past an axial flow hydrokinetic turbine in a fully-developed turbulent flow in an open channel using: turbine-geometry resolving LES (LES-TG) and LES-AD and LES-AL parameterizations. We show that LES-TG reveals very complex large-scale dynamics in the near wake, driven by the interaction of a counter-rotating to the turbine hub vortex and the top-tip shear layer, which appears to influence both the mean flow characteristics and the intensity of wake meandering several rotor diameters downstream. The LES-AD and LES-AL results cannot capture the geometry-induced complex near wake phenomena and yield flows that exhibit important differences with the LES-TG results in the far wake. The mechanisms that give rise to and modeling implications of these differences will be discussed.

[1]This work was supported by Department of Energy DOE (DE-EE0002980 and DE-EE0005482) and Xcel Energy through the Renewable Development Fund (grant RD3-42). Computational resources were provided by the University of Minnesota Supercomputing Institute.

5:11PM E13.00003 Large-eddy simulation of the flow over a hydrokinetic turbine mounted on an erodible bed[1] , XIAOLEI YANG, ALI KHOSRONEJAD, St. Anthony Falls Laboratory, College of Science and Engineering, University of Minnesota, FOTIS SOTIROPOULOS, St. Anthony Falls Laboratory, College of Science and Engineering, Department of Civil Engineering, University of Minnesota — Marine and hydrokinetic (MHK) energy comprises an important source of clean and renewable energy. The beds of natural waterways are usually erodible. The hydrokinetic turbines affect the sediment transport, which, on the other hand, also influences the performance of hydrokinetic turbines. A powerful computational framework for simulating marine and hydrokinetic (MHK) turbine arrays mounted in complex river bathymetry with sediment transport has been developed and validated by our group. In this work we apply this method to simulate the turbulent flow over a hydrokinetic turbine mounted in an open channel with erodible bed. Preliminary results show qualitatively good agreement with the experiment. Detailed comparison with measurements and analysis of the simulation results will be presented in the conference.

[1]This work was supported by the University of Minnesota Initiative for Renewable Energy and the Environment IREE (grant no RO-0004-12). Computational resources were provided by the University of Minnesota Supercomputing Institute.

5:24PM E13.00004 Numerical Modeling and Experimental Analysis of Scale Horizontal Axis Marine Hydrokinetic (MHK) Turbines[1] , TEYMOUR JAVAHERCHI, NICK STELZENMULLER, University of Washington, JOSEPH SEYDEL, The Boeing Company, ALBERTO ALISEDA, University of Washington — We investigate, through a combination of scale model experiments and numerical simulations, the evolution of the flow field around the rotor and in the wake of Marine Hydrokinetic (MHK) turbines. Understanding the dynamics of this flow field is the key to optimizing the energy conversion of single devices and the arrangement of turbines in commercially viable arrays. This work presents a comparison between numerical and experimental results from two different case studies of scaled horizontal axis MHK turbines (45:1 scale). In the first case study, we investigate the effect of Reynolds number (Re=40,000 to 100,000) and Tip Speed Ratio (TSR=5 to 12) variation on the performance and wake structure of a single turbine. In the second case, we study the effect of the turbine downstream spacing (5d to 14d) on the performance and wake development in a coaxial configuration of two turbines. These results provide insights into the dynamics of Horizontal Axis Hydrokinetic Turbines, and by extension to Horizontal Axis Wind Turbines in close proximity to each other, and highlight the capabilities and limitations of the numerical models. Once validated at laboratory scale, the numerical model can be used to address other aspects of MHK turbines at full scale.

[1]Supported by DOE through the National Northwest Marine Renewable Energy Center.

5:37PM E13.00005 Simulation of Marine Hydrokinetic Turbines in Unsteady Flow using Vortex Particle Method , DANNY SALE, ALBERTO ALISEDA, University of Washington — A vortex particle method has been developed to study the performance and wake characteristics of Marine Hydrokinetic turbines. The goals are to understand mean flow and turbulent eddy effects on wake evolution, and the unsteady loading on the rotor and support structures. The vorticity-velocity formulation of the Navier-Stokes equations are solved using a hybrid Lagrangian-Eulerian method involving both vortex particle and spatial mesh discretizations. Particle strengths are modified by vortex stretching, diffusion, and body forces; these terms in the vorticity transport equation involve differential operators and are computed more efficiently on a Cartesian mesh using finite differences. High-order and moment-conserving interpolations allow the particles and mesh to exchange field quantities and particle strengths. An immersed boundary method which introduces a penalization term in the vorticity transport equations provides an efficient way to satisfy the no-slip boundary condition on solid boundaries. To provide further computational speedup, we investigate the use of multicore processors and graphics processing units using the OpenMP and OpenCL interfaces within the Parallel Particle-Mesh Library.

Sunday, November 24, 2013 4:45PM - 6:03PM –

Session E14 Experimental Techniques III: Pressure Sensitive Paint 302 - James Gregory, Ohio State University

4:45PM E14.00001 Identification of the resonant modes in supersonic impinging jets using fast response pressure sensitive paint , TIMOTHY DAVIS, ADAM EDSTRAND, FARRUKH ALVI, LOUIS CATTAFESTA, Florida State University, DAISUKE YORITA, KEISUKE ASAI, Tohoku University — High speed impinging jets have been the focus of several studies owing to their practical application and resonance dominated flow-field. The current study utilizes fast-response pressure sensitive paint (PSP) to examine the jet instability modes of a Mach 1.5 normally impinging jet. These modes are associated with high amplitude, discrete peaks in the power spectra and can be identified as having either axisymmetric or azimuthal modes. Phase-averaged images are acquired at various nozzle to plate spacings and at frequencies of several kHz. Using an unsteady pressure transducer on the impingement surface as a reference signal, a high speed LED with a wavelength of 460 nm is used to illuminate the PSP. The paint fluorescence is then recorded with a CCD camera. The average pressure distribution is removed from the acquired images, resulting in the phase-averaged unsteady pressure distribution. The processed images reveal axisymmetric modes for all nozzle to plate spacings tested except at 4 jet diameters. At this spacing, three distinct resonant modes are identified.

4:58PM E14.00002 Simultaneous pressure and velocity measurements using multi-dye microspheres implementing an intensity-ratio method , DANIEL LACROIX, University of Washington, JULIAN MASSING, Technische Universitaet Berlin, CHRISTIAN CIERPKA, Bundeswehr University Munich, GUO-SHI LI, DANA DABIRI, GAMAL KHALIL, University of Washington, UNIVERSITY OF WASHINGTON TEAM, TECHNISCHE UNIVERSITAET BERLIN COLLABORATION, BUNDESWEHR UNIVERSITY MUNICH COLLABORATION — Luminescent pressure and temperature sensitive paints have been used to quantify pressure and temperature on surfaces, but not in flow fields. By incorporating pressure-sensitive, temperature-sensitive, and reference (non-sensitive) dyes into microspheres, which are then seeded into flow and imaged using high sensitivity EMCCDs, it will be possible to obtain pressure and temperature simultaneously. The pressure and temperature measurement methodologies are coupled with existing particle image velocimetry (PIV) methods to obtain simultaneous pressure, temperature, and velocity. Current research is focused on pressure and velocity measurements of these aerosolized microspheres using multiple cameras and calculating the ratio between the imaged pressure-sensitive and reference intensities. Detailed post-processing, which shows that the standard deviation of the ratios are reduced from approximately 10% to 5-6% will be discussed. In addition, we will discuss the implementation of the intensity-ratio method as well as the calibration process for determining pressure.

5:11PM E14.00003 Pressure and velocity field measurements of pulsating flow in a square channel y-junction , MARKUS PASTUHOFF, ATHANASIA KALPAKLI, P. HENRIK ALFREDSSON, CCGEx, KTH Mechanics — The pressure and velocity fields in a y-junction of a square (40×40 mm^2) cross-section channel were investigated during pulsating flow. One of the sides of the channel was covered with fast responding pressure sensitive paint (PSP) whereas the velocity field at the channel center parallel to the PSP surface was measured using particle image velocimetry (PIV). The flow conditions, in terms of mass flow rate and pulsation frequency, were selected to resemble the flow inside an exhaust manifold of a small internal combustion engine, although the gas was at room temperature. The mass flow was varied between 10 and 130 g/s with pulsations between 0 and 80 Hz. For both the PSP and the PIV measurements images were acquired unsynchronized to the pulses using a high-speed camera and phase averages were formed *a posteriori*. The use of PSP together with PIV demonstrates how the two techniques can be used to verify and complement each other, PIV excelling at the lower mass flow rates and PSP at the higher. It is shown that the signal-to-noise ratio for PSP at low velocities can be enhanced using a technique based on singular value decomposition.[1]

[1]Pastuhoff, M., Yorita, D., Asai, K. & Alfredsson, P.H. 2013 *Meas. Sci. Technol.* **24**, 075301.

5:24PM E14.00004 Temperature Cancellation Method of Motion-Capturing Pressure-Sensitive Paint System , HIROTAKA SAKAUE, JAXA, YUKI YAMADA, TAIKA OKABE, TAKESHI MIYAZAKI, The University of Electro-Communications — Motion-capturing pressure-sensitive paint system uses two luminescent outputs to extract the pressure field on an aerodynamic object. This uses a luminescent imaging technique to relate the luminescent output to the pressure. In the previous study, this system is applied to capture the time-resolved unsteady pressure fields on a fluttering airfoil, and a bullet-shaped model. Pressure-sensitive paint (PSP) has a temperature dependency, which is a major error source for the PSP measurement. Motion-capturing PSP system also involves the temperature dependency of PSP itself. In the presentation, we propose a temperature-cancellation method of the motion-capturing PSP system. This method does not require a separate temperature measurement for the temperature correction that is advantage for capturing the pressure field on a moving object.

5:37PM E14.00005 Characterizations of Inorganic Electro-Luminescence as an Excitation Source of Pressure-Sensitive Paint Measurement System , YOSHIMI IIJIMA, HIROTAKA SAKAUE, JAXA — Electro-luminescence based pressure-sensitive paint (EL-PSP) system uses an inorganic EL as an excitation source for a PSP measurement. It can be directly applied onto a PSP model to eliminate a remote illumination, and gives a uniform illumination on a PSP model without moving/re-directing the illumination. The temperature dependency of the EL-PSP system can be reduced by the opposite temperature dependency of the EL and PSP. An inorganic EL needs an AC input for illumination that creates a periodic excitation of a PSP. It is necessary to characterize the periodic illumination of the EL in terms of a PSP excitation source. At present, it is found that a single pulse of the EL is dependent on the temperature but independent of the pressure. In the presentation, we discuss further the characterizations of the EL as an illumination source of PSP. These include the frequency of the EL illumination and the illumination pattern.

5:50PM E14.00006 Surface Pressure Measurements from Multiline Single-Component Molecular Tagging Velocimetry[1] , DAVID A. OLSON, AHMED M. NAGUIB, MANOOCHEHR M. KOOCHESFAHANI, Michigan State University — This study considers the feasibility of estimating the surface pressure distribution based on high-resolution single-component molecular tagging velocimetry. This approach would be helpful in situations where it is not practical to instrument a surface with pressure taps/sensors. The method relies on the connection between the surface pressure gradient and the second order wall-normal derivative of the velocity tangent to the wall. We show the application of this approach to measuring the surface pressure distribution on the circular cylinder in cross flow at Re = 6,000. Results compare favorably with data in the literature.

[1]This work was supported by AFOSR grant number FA9550-10-1-0342.

Sunday, November 24, 2013 4:45PM - 5:50PM –

Session E16 Biofluids: Physiological III - Experimental Studies in Cardiovascular Flows 304 - Sahar Hendabadi, Illinois Institute of Technology

4:45PM E16.00001 Effect of mitral orifice shape on intra-ventricular filling fluid dynamics[1] , IKECHUKWU OKAFOR, YAGNA ANGIRISH, AJIT YOGANATHAN, Georgia Institute of Technology & Emory University, ARVIND SANTHANAKRISHNAN, Oklahoma State University — The natural geometry of the mitral orifice is D-shaped. However, most current designs of prosthetic valves employ O-shaped orifice geometry. The goal of this study was to compare the effect of geometrical modification between the D and O orifice on the intra-ventricular fluid dynamics during diastolic filling. The different mitral orifice geometries were incorporated into an in vitro left heart simulator consisting of a flexible-walled anatomical left ventricle (LV) physical model enclosed in an acrylic housing. Physiological flow rates and pressures were obtained via tuning systemic resistance and compliance elements in the flow loop. A programmable piston pump was used to generate the LV model wall motion. 2D Particle image velocimetry measurements were conducted along multiple longitudinal planes perpendicular to the annulus plane. During peak diastole, the incoming jet width at the LV central plane was smaller for the D-orifice than that of the O-orifice. Further, the core of the vortex ring in the D-orifice was reduced in size compared to that of the O-orifice. The spatiotemporal spreading of the inflow jet as well as the propagation of the vortex ring will be discussed.

[1]This study was funded by a grant from the National Heart, Lung and Blood Institute (RO1HL70262).

4:58PM E16.00002 Effect of varying heart rate on intra-ventricular filling fluid dynamics[1] , ARVIND SANTHANAKRISHNAN, Oklahoma State University, IKECHUKWU OKAFOR, YAGNA ANGIRISH, AJIT YOGANATHAN, Georgia Institute of Technology & Emory University — Impaired exercise tolerance is used to delineate asymptomatic patients during the clinical diagnosis of diastolic left heart failure. Examining the effects of varying heart rate on intra-ventricular filling can provide a physical understanding of the specific flow characteristics that are impacted during exercise. In this study, diastolic filling was investigated with an anatomical left ventricle (LV) physical model under normal heart rate of 70 bpm, and varying exercise conditions of 100 bpm and 120 bpm. The LV model was incorporated into a flow loop and tuned for physiological inflow rates and outflow pressures. 2D PIV measurements were conducted along 3 parallel longitudinal planes. The systemic pressure was maintained the same across all test conditions. The E/A ratio was maintained within 1.0-1.2 across all heart rates. The strength of the mitral vortex ring formed during E-wave, as well as the peak incoming jet velocity, decreased with increasing heart rate. During peak flow of the A-wave, the vortex ring propagated farther into the LV for 120 bpm as compared to 70 bpm. The results point to the heightened role of the atrial kick for optimal LV filling during exercise conditions.

[1]This study was funded by a grant from the National Heart, Lung and Blood Institute (RO1HL70262).

5:11PM E16.00003 ABSTRACT WITHDRAWN –

5:24PM E16.00004 Assessment of transmitral flow after mitral valve edge-to-edge repair using High-speed particle image velocimetry , MORTEZA JEYHANI, Mech. Eng. Dept., Concordia Univ., Montreal, Canada, SHAHROKH SHAHRIARI, University of Montreal Hospital Research Center (CRCHUM), Montreal, Canada, MICHEL LABROSSE, Mech. Eng. Dept., University of Ottawa, Ottawa, Canada, LYES KADEM, Mech. Eng. Dept., Concordia Univ., Montreal, Canada — Approximately 500,000 people in North America suffer from mitral valve regurgitation (MR). MR is a disorder of the heart in which the mitral valve (MV) leaflets do not close securely during systole. Edge-to-edge repair (EtER) technique can be used to surgically treat MR. This technique produces a double-orifice configuration for the MV. Under these un-physiological conditions, flow downstream of the MV forms a double jet structure that may disturb the intraventricular hemodynamics. Abnormal flow patterns following EtER are mainly characterized by high-shear stress and stagnation zones in the left ventricle (LV), which increase the potential of blood component damage. In this study, a custom-made prosthetic bicuspid MV was used to analyze the LV flow patterns after EtER by means of digital particle image velocimetry (PIV). Although the repair of a MV using EtER technique is an effective approach, this study confirms that EtER leads to changes in the LV flow field, including the generation of a double mitral jet flow and high shear stress regions.

5:37PM E16.00005 Flow Behavior in the Left Heart Ventricle Following Apico-Aortic Bypass Surgery , SHAHROKH SHAHRIARI, University of Montreal Hospital Research Center (CRCHUM), MORTEZA JEYHANI, Concordia University, MICHEL LABROSSE, University of Ottawa, LYES KADEM, Concordia University — Apico-aortic bypass (AAB) surgery is an alternative for transcatheter aortic valve implantation (TAVI) to reduce left ventricle (LV) overload in patients with severe aortic stenosis (AS). It consists in connecting the apex of the LV to the descending thoracic aorta with a valved conduit. Postoperative flow assessments show that two thirds of the outflow is conducted from the LV apex to the conduit, while only one third crosses the native aortic valve. In this study, we performed high speed particle image velocimetry (PIV) measurements of flow pattern within an in vitro elastic model of LV in the presence of a very severe AS, before and after AAB. Results indicate that AAB effectively relieves the LV outflow obstruction; however, it also leads to abnormal ventricular flow patterns. Normal LV flow dynamics is characterized by an emerging mitral jet flow followed by the development of a vortical flow with velocities directed towards the aortic valve, while measurements in the presence of AAB show systolic flow bifurcating to the apical conduit and to the aortic valve outflow tract. This study provides the first insight into the LV flow structure after AAB including outflow jets and disturbed stagnation regions.

Sunday, November 24, 2013 4:45PM - 5:50PM –

Session E17 Biofluids: Locomotion III - Flying 305 - Iman Borazjani, University at Buffalo

4:45PM E17.00001 Lift enhancement in flying snakes , ANUSH KRISHNAN, Boston University, JOHN SOCHA, Virginia Tech, PAVLOS VLACHOS, Virginia Tech, Purdue University, LORENA BARBA, Boston University, George Washington University — Flying snakes use a unique method of aerial locomotion: they jump from tree branches, flatten their bodies and undulate through the air to produce a glide. The shape of their body cross-section during the glide plays an important role in generating lift. We present a computational investigation of the aerodynamics of the cross-sectional shape. We performed two-dimensional simulations of incompressible flow past the anatomically correct cross-section of the species Chrysopelea paradisi, which show that a significant enhancement in lift appears at an angle of attack of 35 degrees, for Reynolds numbers 2000 and above. Previous experiments on physical models also demonstrated an increased lift and at the same angle of attack. The simulations point to the lift enhancement arising from the early separation of the boundary layer on the dorsal surface of the snake profile, without stall. The separated shear layer rolls up and interacts with secondary vorticity in the near-wake, inducing the primary vortex to remain closer to the body and thus cause enhanced suction, resulting in higher lift. In physical experiments, the flow is inherently 3-D due to fluid instabilities, and it is intriguing that the enhanced lift also appears in the two-dimensional simulations.

4:58PM E17.00002 On the hydrodynamics of ray-like swimming[1] , RICHARD G. BOTTOM II, IMAN BORAZJANI, University at Buffalo, ERIN BLEVINS, GEORGE V. LAUDER, Harvard University — There are substantial differences in body shape and motion of stingrays relative to other fish, which drastically affect the hydrodynamics of locomotion. Discovering the flow physics of ray-like locomotion is invaluable not only from a biological standpoint but also for practical application in the development of novel, bio-inspired, man-made vehicles. Here we first develop an analytical model for the stingray's body and fin motion based on experimental laser scan of body shape in the freshwater stingray Potamotrygon orbignyi, and on experimental 3D kinematic data of the wing and body surface obtained from freely-swimming stingrays. The accurate model for the stingray motion is constructed by Fourier analysis of the experimental data resulting in a traveling wave equation with an amplitude coefficient, which is spatially dependent across the fin. Based on this model, we carry out large eddy simulations of the stingray using the immersed boundary method, i.e., the motion of the stingray body is prescribed based on the model, and the motion of the center of mass is calculated. We validate our simulations against experimental data. The simulations reveal the 3D structure of the wake and quantify the swimming performance under different conditions.

[1]This work was partly supported by the Center for Computational Research (CCR), University at Buffalo.

5:11PM E17.00003 Experimental study of the fluid structure interaction for falling cards , RUIJUN TIAN, COULTON SADLER, FANGJUN SHU, New Mexico State University — In this experimental study, high-speed visualization and PIV measurements were conducted to investigate the dynamic evolution of the flow field generated by gravity driven falling cards. Experiments were done in both water and air using glass slides, which are transparent to avoid laser blockage. Fluttering motion (card sliding from side to side while descending) was observed in water while tumbling motion (card rotates w.r.t. it long axis) happened in air. High-speed images of the falling cards were acquired and processed to analyze its kinematics including velocities and accelerations, both translational and rotational. From the card kinematics, the instantaneous fluid dynamic forces/torque were derived, they were related to the surrounding flow field measured using PIV. It is found that the leading edge vortex plays an important role in falling mode. Its evolution and shedding is closely related to the change of translational and angular acceleration of the falling plates, thus influence the falling mode. In the same fluid, a narrow card intends to tumble while a wide card intends to flutter, mainly due to increased moment of inertia. An empirical or theoretical theory is to be developed to predict the motion and trajectory of a gravity driven falling card.

5:24PM E17.00004 An elastic body impacting the water surface; inspired by diving birds , SUNGHWAN JUNG, ALEX OCHS, SEAN GART, Department of Engineering Science and Mechanics, Virginia Tech — We investigate how a soft elastic body responds to water-entry impact analogous to a bird diving into water to catch prey. Dumbbell shaped objects made of two acrylic spheres connected by an elastic rod are dropped into water. A buckling threshold was found by varying impact force and elastic rod stiffness. This threshold may have implication as to how birds are able to safely dive into water at high speeds and avoid any neck-injury.

5:37PM E17.00005 Could flapping foil propulsion become a commercial shipping reality?[1] , BRENDEN EPPS, Dartmouth College — Since the 1990s, fluid dynamicists have made great strides towards understanding the physics of flapping foil propulsion. However, to date, few commercial shipping vessels employ this technology. Why? How does the efficiency of a flapping hydrofoil compare to that of a conventional screw propeller? What technical challenges exist, and how might we overcome them in order to make flapping foil propulsion a reality for commercial shipping vessels?

[1]This work is supported by Walvisstaart Exploitatie BV

Sunday, November 24, 2013 4:45PM - 6:03PM –

Session E18 Flow Visualization 306/307 - John Wettlaufer, University of Oxford and Yale University

4:45PM E18.00001 Seminar in Flow Visualization at Lafayette College: Variations on the Hertzberg Effect[1] , JENN STROUD ROSSMANN, Lafayette College — Flow visualization reveals an invisible world of fluid dynamics, blending scientific investigation and artistic exploration. The resulting images have inspired, and in some cases themselves become appreciated as, art. At Lafayette College, a sophomore-level seminar in *The Art and Science of Flow Visualization* exposes students to these techniques and the science of fluid mechanics, and to the photographic methods needed to create effective images that are successful both scientifically and artistically. Unlike other courses in flow visualization, this course assumes no a priori familiarity with fluid flow or with photography. The fundamentals of both are taught and practiced in a studio setting. Students are engaged in an interdisciplinary discourse about fluids and physics, photography, scientific ethics, and historical societal responses to science and art. Relevant texts from several disciplines are read, discussed, and responded to in student writing. This seminar approach makes flow visualization and fluid dynamics a natural part of a liberal education. The development, implementation, and assessment of this team-taught course at Lafayette College will be discussed.

[1]Support provided by National Science Foundation

4:58PM E18.00002 Student designed experiments to learn fluids[1] , CATALINA STERN, Facultad de Ciencias, UNAM — Lasers and high speed cameras are a wonderful tool to visualize the very complex behavior of fluids, and to help students grasp concepts like turbulence, surface tension and vorticity. In this work we present experiments done by physics students in their senior year at the School of Science of the National University of Mexico as a final project in the continuum mechanics course. Every semester, the students make an oral presentation of their work and videos and images are kept in the web page "Pasión por los Fluidos".

[1]I acknowledge support from the Physics Department of Facultad de Ciencias, Universidad Nacional Autónoma de México

5:11PM E18.00003 Designing and Creating a Set of New Lab Experiments for a Traditional Fluid Mechanics Course in Civil Engineering , DAN BUDNY, University of Pittsburgh — Many fluids lab facilities and their associated student experiences were built back in the 1960-1970 time frames. They typically consisted of large facilities that included wind tunnels, flumes, wet wells, pump stations, etc. Today these labs are physically and pedagogically out dated and the need for lab space is forcing the closing of large scale labs. This is the same basic problem within the Swanson School of Engineering at the University of Pittsburgh. Thus we have replaced all the old equipment and lab experiences with small bench top experiments with a focus on applying the large body of knowledge associate with better student learning experiences. This paper will describe the concepts behind the design of the new experiments and the learning improvements discovered as a result of moving from a few large experiments to a larger number of smaller scale experiments.

5:24PM E18.00004 That is Cool: the Nature Of Aesthetics in Fluid Physics[1] , JEAN HERTZBERG, University of Colorado, Boulder — Aesthetics has historically been defined as the study of beauty and thus as a metric of art. More recently, psychologists are using the term to describe a spectrum of responses from "I hate it" to "I love it." In the context of fluid physics, what is beautiful? What elicits a "Wow! Awesome! Cool!" response versus a snore? Can we use aesthetics to deepen or change students' or the public's perceptions of physics and/or the world around them? For example, students seem to appreciate the aesthetics of destruction: environmental fluid dynamics such as storms, tornadoes, floods and wildfires are often responsible for massive destruction, yet humans draw pleasure from watching such physics and the attendant destruction from a safe distance. Can this voyeurism be turned to our advantage in communicating science? Observations of student and Facebook Flow Visualization group choices for fluid physics that draw a positive aesthetic response are sorted into empirical categories; the aesthetics of beauty, power, destruction, and oddness. Each aesthetic will be illustrated with examples drawn from flow visualizations from both the Flow Visualization course (MCEN 4151) taught at the University of Colorado, Boulder, and sources on the web.

[1]This work is supported by NSF: EEC 1240294.

5:37PM E18.00005 F* Yeah Fluid Dynamics: Lessons from online outreach** , NICOLE SHARP, Texas A&M University — The fluid dynamics education outreach blog FYFD features photos, videos, and research along with concise, accessible explanations of phenomena every weekday. Over the past three years, the blog has attracted an audience of roughly 200,000 online followers. Reader survey results indicate that over half of the blog's audience works or studies in non-fluids fields. Twenty-nine percent of all survey respondents indicate that FYFD has been a positive influence on their desire to pursue fluid dynamics in their education or career. Of these positively influenced readers, over two-thirds have high-school or undergraduate-level education, indicating a significant audience of potential future fluid dynamicists. This talk will utilize a mixture of reader metrics, web analytics, and anecdotal evidence to discuss what makes science outreach successful and how we, as a community, can benefit from promoting fluid dynamics to a wider audience. *http://tinyurl.com/azjjgj2*

5:50PM E18.00006 Visualization of Tensor Quantities Used in Computational Turbulent Combustion , TIMOTHY LUCIANI, ADRIAN MARIES, G. ELISABETA MARAI, Department of Computer Science, University of Pittsburgh, MEHDI B. NIK, Department of Mechanical Engineering and Materials Science, University of Pittsburgh, SERVER L. YILMAZ, University of Pittsburgh — Simulation and modeling of turbulent flow, and of turbulent reacting flow in particular, involves solving for and analyzing time-dependent and spatially dense tensor quantities, such as turbulent stress tensors. The interactive visual exploration of these tensor quantities can effectively steer the computational modeling of combustion systems. In this work we present an integrated, hybrid tensor visualization tool. Volume renderings of the 3D quantities reveal the structure of the dataset, allowing users to identify regions of interest. The regions of interest can then be further explored using the 2D glyph-based representations. Finally, streamlines are also provided to help reveal the structure of the flow through the region of interest. In order to facilitate time-space analysis, we provide a small-multiples interface: users can compare thumbnail, low-resolution representations of different time steps and then focus on one time step to view in detail. We demonstrate our application on several turbulent configurations, and show this approach can successfully capture and highlight numerical artifacts in the underlying computational model.

Corresponding Author: T. Luciani, Tel: (412) 624-8460. Fax: (412) 624-8854. E-mail: tbl8@cs.pitt.edu.

Sunday, November 24, 2013 4:45PM - 6:03PM –

Session E19 Rarefied Gases and DSMC 310/311 - Deborah Levin, Pennsylvania State University

4:45PM E19.00001 Study of shock-shock interactions for a Double Wedge using the DSMC Approach[1] , DEBOAH LEVIN, VARUN PATIL, Penn State University, SERGEY GIMELSHEIN, University of Southern California, JOANNA AUSTIN, University of Illinois, Urbana-Champaigne — The Direct Simulation Monte Carlo (DSMC) method, an approach for modeling finite-Knudsen number flows is being used to study the laminar, shock-shock interactions from hypersonic flows about a double-wedge configuration in the Hypervelocity Expansion Tube (HET) facility. The study focuses on the investigation of Mach 7 nitrogen flows about a 30-/55-deg double wedge model for stagnation enthalpy of 8.0 MJ/kg. Schlierens are generated to visualize the shock structure and shock-shock interactions present in these flows and are compared with the experimental images. The computed heat transfer values from the simulations match the experiment along the first surface, but on the second wedge the computed heat transfer distribution over predicts the measured peak values. The influence of different models for nonequilibrium nitrogen dissociation, rotational and vibrational relaxation rates, and gas-surface interactions on the shock interaction region are analyzed for high enthalpy flow features and heat transfer rates. Overall good agreement is observed in the experimental and computational results. Studies are being performed related to flow unsteadiness and three-dimensional affects to resolve remaining discrepancies between measurements and modeling.

[1]Air Force Office of Scientific Research through AFOSR Grant No. FA9550-11-1-0129 with a subcontract award number 2010-06171-01 to PSU.

4:58PM E19.00002 Measurement of Tangential Momentum Accommodation Coefficient (TMAC) using a Disc Spin-Down Experiment in Low Pressure Gas , TATHAGATA ACHARYA, JORDAN FALGOUST, MICHAEL MARTIN, Louisiana State University, RICHARD RASMUSSEN, Guidance Dynamics Corporation — The objective is the measurement of TMAC for gas versus surface interactions. An experimental facility is built to accommodate a disc spin-down experiment in various gas pressures. The experiment measures the drag on the surface of the disc through measurement of its rotational speed during spin-down. Computational fluid dynamics is used to determine an acceptable shape and size of the facility and to estimate the pressure at which free molecular flow regime may be reached. The spin-down speed is translated to angular deceleration. Torque is obtained from disc moment of inertia and the angular deceleration. Data shows that the torque is a linear function of angular velocity. Torque is non-dimensionalized and is plotted against Reynolds number (Re). Between atmospheric pressure and a pressure of 357 Pa, the non-dimensional torque decreases with Re. At 2.7 Pa, the non-dimensional torque does not show any change with Re and the system presumably attains continuum breakdown. At a pressure of 0.71 Pa the free molecular flow regime is reached. The measured TMAC between air and aluminum shows the range between 0.7209 and 0.7355. Future work will measure the TMAC of materials commonly used in aerospace systems such as titanium, kapton, and carbon fiber.

5:11PM E19.00003 Maximum-entropy reconstruction method for moment-based solution of the Boltzmann equation , DUSTIN SUMMY, DALE PULLIN, California Institute of Technology — We describe a method for a moment-based solution of the Boltzmann equation. This starts with moment equations for a $10+9N, N=0,1,2...$-moment representation. The partial-differential equations (PDEs) for these moments are unclosed, containing both higher-order moments and molecular-collision terms. These are evaluated using a maximum-entropy construction of the velocity distribution function $f(\mathbf{c},\mathbf{x},t)$, using the known moments, within a finite-box domain of single-particle-velocity ($\mathbf{c}$) space. Use of a finite-domain alleviates known problems (Junk and Unterreiter, *Continuum Mech. Thermodyn.*, 2002) concerning existence and uniqueness of the reconstruction. Unclosed moments are evaluated with quadrature while collision terms are calculated using a Monte-Carlo method. This allows integration of the moment PDEs in time. Illustrative examples will include zero-space- dimensional relaxation of $f(\mathbf{c},t)$ from a Mott-Smith-like initial condition toward equilibrium and one-space dimensional, finite Knudsen number, planar Couette flow. Comparison with results using the direct-simulation Monte-Carlo method will be presented.

5:24PM E19.00004 Shock Structure at Moderate and Large Mach Numbers , SAMUEL PAOLUCCI, CHRISTOPHER PAOLUCCI, University of Notre Dame — The structure of gas-dynamic shock waves at hypersonic conditions is of great interest. The Navier-Stokes formulation is known to yield incorrect shock profiles even at moderate Mach numbers. This is an excellent test problem for extensions of such equations since excellent experimental results are available. Continuum theories, and indeed most statistical mechanics theories, that have been advanced to reconcile such discrepancies have not been fully successful. Here, we present a second order formulation of the governing stress tensor and heat flux based solely on a continuum formulation. The constitutive equations for the gas, in addition to the known transport properties, also introduce additional viscosity and thermal conductivities which generally depend on density and temperature. Their specific dependence are estimated from kinetic theory. Results of the second-order equations applied to the shock structure are obtained for monatomic and diatomic gases over a large range of Mach numbers and are compared to experimental results.

5:37PM E19.00005 Thermal transpiration of a rarefied gas between parallel plates with different accommodation coefficients , TOSHIYUKI DOI, Department of Applied Mathematics and Physics, Tottori University — Thermal transpiration of a rarefied gas between parallel plates with different accommodation coefficients is studied on the basis of the linearized Boltzmann equation for a hard sphere molecular gas. The Boltzmann equation is solved numerically using a finite difference method. The macroscopic variables as well as the mass flow rate of the gas are studied over a wide range of the Knudsen number (the mean free path divided by the channel width) and the two accommodation coefficients. When the Knudsen number is not so small, the mass flow rate of the gas increases as the accommodation coefficients decrease. When the Knudsen number is sufficiently small, however, the tendency is opposite and the mass flow rate basically decreases as the accommodation coefficients decrease. The solution for infinitesimally small accommodation coefficients is also discussed.

5:50PM E19.00006 The role of divergences for shock waves , FRANCISCO URIBE, Universidad Autónoma Metropolitana — Several continuum theories for shock waves give rise to a set of differential equations in which the analysis of the underlying vector field can be done using the tools of the theory of dynamical systems. We illustrate the importance of the divergences associated with the vector field by considering the ideas by Maxwell and Cattaneo and applied them to study shock waves in dilute gases. Different theoretical descriptions for shock waves are mentioned and some of them are compared with experimental data and computer simulations. Our goal is to derive conditions under which the shock wave problem has a solution by analyzing the singularities of the vector field.

Sunday, November 24, 2013 4:45PM - 5:50PM –

Session E20 Boundary Layers IV: Flow through Pipes 315 - Stephen B. Pope, Cornell University

4:45PM E20.00001 Large-eddy simulation of turbulent pipe flow at large Reynolds number , NAMIKO SAITO, DALE PULLIN, California Institute of Technology, HUGH BLACKBURN, Monash University — We describe large-eddy simulations (LES), using a spectral-element method, of turbulent smooth- and rough-wall pipe flows. The spectral-element code SEMTEX was used (Blackburn and Sherwin *J. Comput. Phys.* 2004) in a mode where the axial direction is treated using Fourier modes, with a spectral-element representation within the cross-flow plane with Dirichlet boundary conditions on the circular pipe boundary. The stretched-vortex subgrid-stress model is utilized together with the wall-model of Chung and Pullin (*JFM, 2009*). For rough-wall flows, local subgrid roughness is incorporated by the addition of an 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 is used in both the inner-scaling ansatz for the unsteady term of the wall-normal integration of the stream-wise momentum equation, and also in the log-like profile used to give a boundary condition for the outer-flow LES. Results will be discussed that include variation of the skin-friction coefficient as a function of both Reynolds number and the ratio of k_s to the pipe radius, and also mean velocity profiles and some turbulence statistics.

4:58PM E20.00002 Deconstructing the effectiveness of opposition control in turbulent pipe flow[1] , MITUL LUHAR, California Institute of Technology, ATI S. SHARMA, University of Southampton, BEVERLEY J. MCKEON, California Institute of Technology — We develop a simple model for opposition control based on the resolvent analysis of McKeon & Sharma (2010, *J Fluid Mech*). This model decomposes the velocity field for turbulent pipe flow into a series of highly-amplified response modes, identified via a gain analysis of the Fourier-transformed Navier-Stokes equations. Changing the boundary conditions to reflect control alters the structure and amplification of these velocity responses, such that a reduction in gain signifies a reduction in drag. With basic assumptions, this rank-1 model reproduces trends seen in previous DNS and LES. Further, a wavenumber-frequency breakdown helps explain the deterioration of opposition control performance with increasing sensor elevation and Reynolds number. We show that opposition control only suppresses attached modes localized near the wall; detached modes, which are more energetic at higher Reynolds number and active far from the wall, are further amplified. Such detached modes require a phase lag between sensor and actuator velocity for suppression. Thus, the efficacy of traditional opposition control is determined by a tradeoff between modes sensed, but it may be possible to prescribe an optimal scheme tailored to individual mode behavior.

[1]This work was supported by AFOSR grant FA9550-12-1-0469 (ML, BJM).

5:11PM E20.00003 Mechanism for skin friction reduction in temporally accelerated turbulent pipe flow[1] , JAE HWA LEE, RONALD J. ADRIAN, Arizona State University — Direct numerical simulations of temporally accelerating turbulent pipe flow are performed to examine the modification of the coherent structures due to acceleration and its relationship to the reduction of turbulent skin friction. Two types of simulations are performed: a) fully developed turbulent flow subjected to constant mean acceleration, and b) evolution of a single hairpin eddy subjected to the same acceleration. The initial eddies are extracted by conditional averaged flow fields associated with second-quadrant Reynolds shear stress events from DNS data of the fully developed turbulent pipe flow at the initial Reynolds number. In the case of fully turbulent initial flow, the temporal acceleration increases the Reynolds number from Re_D =5,300 to 26,500, and the response of the turbulence is found to be delayed relative to the response of the mean flow, as also reported by previous studies. The delay causes the ratio of velocity induced by the hairpin to the mean velocity to decrease below the threshold value for nonlinear formation of new hairpin vortices from the initial hairpin. The autogeneration of new hairpin vortices is suppressed, resulting in reduction of turbulent transport and, consequently, reduction of skin friction.

[1]This research was supported by NSF-CBET Award 1335731.

5:24PM E20.00004 Distinct organisational states of large-scale motions in turbulent pipe flow , FRANCESCA M. SOGARO, DAVID J.C. DENNIS, University of Liverpool — An experimental investigation focussing on the structural organisation of large scale motions (LSMs) in fully developed turbulent pipe flow has been conducted in the Very Large Scale Pipe Flow (VLSPF) facility at the University of Liverpool. Measurements using high-speed stereoscopic particle image velocimetry in the radial-azimuthal plane at a Reynolds number of 35000 (based on bulk velocity and pipe diameter) have been analysed, paying particular attention to the two-point spatial correlation of the streamwise velocity fluctuations. A method has been developed to select similar individual instances (in time) of the planar correlation to identify the presence of a set of distinct organisational states. The application of the selective correlation method demonstrates how the pairs of positive-negative correlation directly correspond to the presence of persistent positive and negative streamwise velocity fluctuations in the instantaneous velocity fields, often referred to as LSMs. The duration that the selective spatial correlation indicates the flow is in a certain state can be related to the length of the large scale motions in the flow and provides a method of identifying and quantifying the characteristics of LSMs in turbulent pipe flow.

5:37PM E20.00005 Statistics and large scales in turbulent pipe and channel flows[1] , JIN LEE, KAIST, JAE HWA LEE, Arizona State University, HYUNG JIN SUNG, KAIST — Turbulent pipe and channel flows have been generally accepted as having a similar turbulent flow structure. However, turbulence statistics in the core region of pipe flow are different from that of channel flow owing to the difference in the averaged spanwise dimension of the low-speed structures. In particular, wall-normal and spanwise stresses of channel flow are smaller than those of pipe flow. In the present study, DNS dataset of turbulent channel and pipe flows with the friction Reynolds number Re=934 have been compared to elucidate the difference of statistics in terms of the populations of large and very-large scales in the low-speed region. To this end, large and very-large scales were extracted by a low-pass filtered streak detection algorithm. We found that the population density of large scales of pipe flow is more increased in the core region than that of channel flow. Although the density of very-large scales of pipe flow decreases, the area of low-speed region increases due to the large number of large scales. Further comparison of pipe and channel flows showed that the higher turbulence intensity of pipe flow is caused by the interference of large scales with the azimuthal distance.

[1]This study was supported by the Creative Research Initiatives program (No. 2013-0003364) of the National Research Foundation of Korea (MSIP) and was partially supported by KISTI under the Strategic Supercomputing Support Program.

Sunday, November 24, 2013 4:45PM - 5:50PM –

Session E21 Biofluids: Physiological IV - Experimental Studies in Respiratory Flows 316 - Daniele Schiavazzi, University of California, San Diego

4:45PM E21.00001 Liquid Therapy Delivery Models Using Microfluidic Airways , MOLLY K. MULLIGAN, Technion - Israel Institute of Technology, Haifa, Israel, JAMES B. GROTBERG, University of Michigan, Ann Arbor MI, US, DAN WAISMAN, Technion - Israel Institute of Technology, Haifa, Israel, MARCEL FILOCHE, Ecole Polytechnique, Paris, France, JOSUÉ SZNITMAN, Technion - Israel Institute of Technology, Haifa, Israel — The propagation and break-up of viscous and surfactant-laden liquid plugs in the lungs is an active area of research in view of liquid plug installation in the lungs to treat a host of different pulmonary conditions. This includes Infant Respiratory Distress Syndrome (IRDS) the primary cause of neonatal death and disability. Until present, experimental studies of liquid plugs have generally been restricted to low-viscosity Newtonian fluids along a single bifurcation. However, these fluids reflect poorly the actual liquid medication therapies used to treat pulmonary conditions. The present work attempts to uncover the propagation, rupture and break-up of liquid plugs in the airway tree using microfluidic models spanning three or more generations of the bronchiole tree. Our approach allows the dynamics of plug propagation and break-up to be studied in real-time, in a one-to-one scale *in vitro* model, as a function of fluid rheology, trailing film dynamics and bronchial tree geometry. *Understanding these dynamics are a first and necessary step to deliver more effectively boluses of liquid medication to the lungs* while minimizing the injury caused to epithelial cells lining the lungs from the rupture of such liquid plugs.

4:58PM E21.00002 Acinus-on-a-chip: a microfluidic platform for pulmonary acinar flows , RAMI FISHLER, MOLLY MULLIGAN, JOSUE SZNITMAN, Technion- Israel Institute of technology, SZNITMAN BIOFLUIDS TEAM — Convective respiratory flows in the pulmonary acinus and their influence on the fate of inhaled particles are typically studied using computational fluid dynamics (CFD) or scaled-up experimental models. However, current experiments generally capture only flow dynamics, without inhaled particle dynamics, due to difficulties in simultaneously matching flow and particle dynamics. In an effort to overcome these limitations, we have designed a novel microfluidic device mimicking acinar flow conditions directly at the physiological scale. The model features an anatomically-inspired acinar geometry with five dichotomously branching airway generations lined with periodically expanding and contracting alveoli. Using micro-particle image velocimetry (PIV), we reveal experimentally a gradual transition of alveolar flow patterns along the acinar tree from recirculating to radial streamlines, in support of previous predictions from CFD simulations. We demonstrate the applicability of the device for studying the mechanisms of particle deposition in the pulmonary acinus by mapping deposition sites of airborne fluorescent micro-particles (0.1-1μm) and visualizing trajectories of airborne incense particles inside the system.

5:11PM E21.00003 Steady Flow in Subject-Specific Human Airways from Mouth to Sixth Bronchial Generation , ANDREW BANKO, FILIPPO COLETTI, Stanford University, DANIELE SCHIAVAZZI, University of California, San Diego, CHRISTOPHER ELKINS, JOHN EATON, Stanford University — Understanding the complex flow topology within the human lung is critical to assess gas exchange and particle transport as they relate to the development and treatment of respiratory diseases. While idealized airway models have been investigated extensively, only limited information is available for anatomically accurate geometries. We have measured the full three-dimensional, mean velocity field from the mouth to the sixth bronchial generation in a patient-specific geometry at steady inspiration. Magnetic resonance velocimetry is used to measure the flow of water at realistic Reynolds number in a 3D-printed model derived from the CT scan of a healthy subject. The canonical laryngeal jet is observed; however, its structure is altered by an upstream jet behind the tongue, which is not discussed in the literature. Regions of separation in the supraglottic space are found to generate streamwise vortices. The resulting swirl persists to the first bifurcation and modifies the vorticity distribution in the main bronchi relative to that of a symmetric bifurcation with uniform inlet conditions. An integral momentum distortion parameter is calculated along several complete bronchial paths to assess the impact of branching angle and generation length on the flow field.

5:24PM E21.00004 Characterization of Ventilatory Modes in Dragonfly Nymph , CHRIS ROH[1], THERESA SAXTON-FOX, MORTEZA GHARIB, California Institute of Technology — A dragonfly nymph's highly modified hindgut has multiple ventilatory modes: hyperventilation (i.e. jet propulsion), gulping ventilation (extended expiratory phase) and normal ventilation. Each mode involves dynamic manipulation of the exit diameter and pressure. To study the different fluid dynamics associated with the three modes, Anisopteran larvae of the family Aeshnidae were tethered onto a rod for flow visualization. The result showed distinct flow structures. The hyperventilation showed a highly turbulent and powerful jet that occurred at high frequency. The gulping ventilation produced a single vortex at a moderate frequency. The normal ventilation showed two distinct vortices, a low-Reynolds number vortex, followed by a high-Reynolds number vortex. Furthermore, a correlation of the formation of the vortices with the movement of the sternum showed that the dragonfly is actively controlling the timing and the speed of the vortices to have them at equal distance from the jet exit at the onset of inspiration. This behavior prevents inspiration of the oxygen deficient expirated water, resulting in the maximization of the oxygen intake.

[1]Supported by NSF GRFP

5:37PM E21.00005 Particle Image Velocimetry Measurements in an Anatomically-Accurate Scaled Model of the Mammalian Nasal Cavity , CHRISTOPHER RUMPLE, MICHAEL KRANE, JOSEPH RICHTER, BRENT CRAVEN, Penn State University — The mammalian nose is a multi-purpose organ that houses a convoluted airway labyrinth responsible for respiratory air conditioning, filtering of environmental contaminants, and chemical sensing. Because of the complexity of the nasal cavity, the anatomy and function of these upper airways remain poorly understood in most mammals. However, recent advances in high-resolution medical imaging, computational modeling, and experimental flow measurement techniques are now permitting the study of respiratory airflow and olfactory transport phenomena in anatomically-accurate reconstructions of the nasal cavity. Here, we focus on efforts to manufacture an anatomically-accurate transparent model for stereoscopic particle image velocimetry (SPIV) measurements. Challenges in the design and manufacture of an index-matched anatomical model are addressed. PIV measurements are presented, which are used to validate concurrent computational fluid dynamics (CFD) simulations of mammalian nasal airflow. Supported by the National Science Foundation.

Sunday, November 24, 2013 4:45PM - 5:50PM –

Session E23 Turbulence: Theory III - Wall-Bounded Flows 318 - Dennice Gayme, Johns Hopkins University

4:45PM E23.00001 Phase relationships and amplitude modulation in wall turbulence[1] , BEVERLEY MCKEON, California Institute of Technology, DANIEL CHUNG, University of Melbourne — We present a framework for predicting the interactions between the large-scale motion and the underlying stress fluctuations in wall turbulence, or the apparent amplitude modulation effect described by, e.g., Mathis et al (Phys. Fluids, 2011). The dynamical equations for stress fluctuations are obtained from a scale decomposition of the governing equations which can be shown to be consistent with the resolvent analysis of McKeon & Sharma (2010). The spatial phase shift between the large-scale motion and stress fluctuations is revealed as being related to critical layer behavior identified therein. Consistent with experiments, the analysis predicts that the zero-crossing height of the amplitude modulation statistic, and corresponding $\pi/2$ lead of the small scales with respect to the large scale identified via cross-correlation techniques, coincides with the wall-normal location of the peak large-scale energetic activity. Simple approximations in the logarithmic region of the mean velocity link the behavior of the amplitude modulation statistic to the wall-normal profiles of the background (mean) turbulent stresses.

[1]The support of AFOSR under grant #FA 9550-09-1-0701 (BJM) is gratefully acknowledged.

4:58PM E23.00002 An Integral Method to Evaluate Wall Heat Flux Suitable For Experimental Data , ALIREZA EBADI, University of New Hampshire, FARAZ MEHDI, Texas Tech University, CHRISTOPHER WHITE, University of New Hampshire — An integral method to evaluate wall heat flux in turbulent boundary layers is presented. The method is mathematically exact and has the advantage of having no explicit streamwise gradient terms, thus making it amenable to experimental data. Using existing data sets, the method is shown to work in both zero- and adverse-pressure gradient boundary layers. The method is particularly useful for the latter case where Reynolds analogy does not hold and the wall heat flux must be measured directly.

5:11PM E23.00003 Universal Karman constant in canonical wall turbulence , ZHEN-SU SHE, XI CHEN, Peking university, FAZLE HUSSAIN, Texas Tech University — A universal Karman constant $\kappa \approx 0.45$ is obtained for all three canonical wall-bounded turbulent flows (channel, pipe and turbulent boundary layer - TBL) for Reynolds numbers (Re) larger than 5,000. A New method for measuring κ from mean velocity profile (MVP) data, reported previously, is applied to 54 sets of recent experimental data (24 for smooth pipe, 8 for rough pipe, 6 for smooth channel and 16 for smooth TBL) and 3 sets of DNS data (2 for smooth channel, 1 for smooth pipe), which uniformly supports the idea that Karman constant is universal, contrary to the recent suggestions that kappa is a function of Re and geometry; its value is almost 10% larger than the classical value of 0.41, with even higher values reported at moderate Re. The validity of the log-law seems to be thus firmly established.

5:24PM E23.00004 Variation approach to describe bulk flow of wall turbulence , XI CHEN, Peking University, FAZLE HUSSAIN, Texas Tech University, ZHEN-SU SHE, Peking University — A mean field theory for the mean velocity profile in the bulk of canonical wall bounded turbulence (channel, pipe and turbulent boundary layer) is developed, in good agreement with empirical data over a wide range of the Reynolds number (Re). In analogy to the Landau's mean field theory (1937) using order parameter to explain phase transition in critical phenomena, the current theory builds a variational description for a characteristic length scale, which minimizes the effective free energy for turbulent momentum flux. It leads to a defect power law for the characteristic length scale, not only offering a novel derivation for the logarithmic mean velocity profile, but also quantifying the geometry effect in turbulent channel and pipe flows. Finally, the Karman constant is proved to be a universal constant under such the variational description, and its physical interpretation is also presented.

5:37PM E23.00005 Nonlinearity and the energy cascade in the resolvent analysis of wall turbulence[1] , ATI SHARMA, University of Southampton, BEVERLEY MCKEON, California Institute of Technology — The resolvent analysis of wall turbulence can be used to characterise velocity response modes derived from a gain analysis of the linear resolvent operator obtained from the Navier-Stokes equations projected into wavenumber-frequency space (k, n, ω), e.g. McKeon & Sharma (JFM, 2010). Simple combinations of response modes that are triadically consistent in (k, n, ω) have been shown to give rise to complex coherent structure, Sharma & McKeon (JFM, 2013), however the selection of these combinations was phenomenologically-driven. In the full analysis, the nonlinear interaction between response modes necessarily gives rise to self-sustaining turbulence. In this paper, we report how the nonlinearity acts to reinforce certain combinations of modes over others, cascades energy between wavenumbers and modes, and determines the relative phase and amplitude of the resolvent response modes.

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

Sunday, November 24, 2013 4:45PM - 6:03PM –

Session E24 Aerodynamics I 319 - Mohamed Gad-el-Hak, Virginia Commonwealth University

4:45PM E24.00001 Flow Structure on a Rotating Wing: Effect of Rossby Number , MAXWELL WOLFINGER, DONALD ROCKWELL, Lehigh University — The flow structure on a rotating wing is determined via stereoscopic particle image velocimetry. Sectional and three-dimensional, volumetric reconstructions define the flow patterns as a function of Rossby number *Ro*. An aspect ratio $AR = 1$ rectangular, flat plate is rotated at a geometric angle of attack $\alpha = 45°$. The flow structure is determined at various angles of rotation, in order to characterize both the initial development and the fully evolved state of the flow structure. The Rossby number $Ro = r_g/C$ is varied via alteration of the radius of gyration r_g of the wing, to give values from $Ro = 1.2$ to $Ro = 5.1$. Large changes of the flow structure are represented by images of of spanwise vorticity, Q-criterion; spanwise velocity; and downwash velocity. At the lowest Rossby number $Ro = 1.2$, a vortex is attached to the leading edge of the wing; it is present along most of the span. At higher Rossby numbers $Ro = 2.1$ and $Ro = 5.1$, this leading-edge vortex becomes less organized and deflects away from the surface of the wing. At a Rossby number $Ro = 5.1$ the structure of the flow in the vicinity of the leading edge resembles a separated shear layer. The nature of other elements of the three-dimensional flow, such as the root and tip vortices and the downwash velocity, are closely related to the degree of coherence of the leading-edge vortex.

4:58PM E24.00002 Vortex Interaction on Low Aspect Ratio Membrane Wings , RYE M. WALDMAN, KENNETH S. BREUER, Brown University — Inspired by the flight of bats and by recent interest in Micro Air Vehicles, we present measurements on the steady and unsteady behavior of low aspect ratio membrane wings. We conduct wind tunnel experiments with coupled force, kinematic, and flow field measurements, both on the wing and in the near wake. Membrane wings interact strongly with the vortices shed from the leading- and trailing-edges and the wing tips, and the details of the membrane support play an important role in the fluid-structure interaction. Membranes that are supported at the wing tip exhibit less membrane flutter, more coherent tip vortices, and enhanced lift. The interior wake can exhibit organized spanwise vortex shedding, and shows little influence from the tip vortex. In contrast, membranes with an unsupported wing tip show exaggerated static deformation, significant membrane fluttering and a diffuse, unsteady tip vortex. The unsteady tip vortex modifies the behavior of the interior wake, disrupting the wake coherence.

5:11PM E24.00003 Unique stability modes of low aspect ratio wings , KAMRAN MOHSENI, MATT SHIELDS, University of Florida — The unique aerodynamic regime of low aspect ratio (LAR) wings is strongly affected by the phenomenon of roll stall. In this study, it is shown that roll stall induces inherently aerodynamic stability modes on a flat plate wing with an aspect ratio of unity. These modes are seen to create divergent oscillations in the lateral state variables even for minor perturbations from equilibrium flight. Furthermore, the nature of the response is fundamentally altered in the presence of angle of attack variations; if the frequency of the angle of attack oscillations is close to the natural frequency of the lateral response, the bank angle ϕ is seen to drift away from equilibrium in a manner not well modeled by a linear stability analysis. This newly considered mode, inherent to LAR wings, is referred to as the roll resonance mode due to its dependence on the frequencies of lateral and longitudinal motion. A linear time invariant model is shown to accurately represent the initial condition response of the pure lateral mode, and a linear time variant model in which the roll stability derivative is updated at every time step captures the divergent response of roll resonance. Understanding these modes is critical for implementation of improved control laws for Micro Aerial Vehicles.

5:24PM E24.00004 Vorticity Confinement Applied to Turbulent Wing Tip Vortices for Wake-Integral Drag Prediction[1] , KRISTOPHER PIERSON, ALEX POVITSKY, The University of Akron — In the current study the vorticity confinement (VC) approach was applied to tip vortices shed by edges of stationary wings in order to predict induced drag by far-field integration in Trefftz plane. The VC parameter was evaluated first by application to convection of vortices in 2-D uniform flow and then to tip vortices shed in 3-D simulation of finite-aspect ratio rectangular wing in subsonic flight. Dependence of VC parameter on the flight Mach number and the angle of attack was evaluated. The aerodynamic drag results with application of VC to prevent numerical diffusion are much closer to analytic lifting line theory compared to integration over surface of wing while the viscous profile drag is more accurately evaluated by surface integration. To apply VC to viscous and turbulent flows, it is shown that VC does not affect the physical rate of dissipation of vortices in viscous/turbulent flows at time scales corresponding to convection of vortices from the wing to Trefftz plane of integration. To account for turbulent effects on tip vortices, VC was applied in combination with Spalart-Allmaras, k-ε, and six Reynolds stresses models of turbulence. The results are compared to experiments to validate the physical dissipation of tip vortex.

[1]This research was supported by The Dayton Area Graduate Studies Institute (DAGSI) and US Air Force Research Laboratory (AFRL) grants in 2009-2013, US Army Research Office (ARO) in 2012-2013 and ASEE/AFRL summer faculty grant.

5:37PM E24.00005 Lattice Boltzmann simulations of deformation and efficiency of a chord-wise flexible wing in a free stream flow , DEWEI QI, Wester Michigan University, GUOWEI HE, LNM, Institute of Mechanics, Chinese Academy of Sciences, Beijing, 100080, China — Flapping of a chord-wise flexible wing in a steady free stream is studied by using a lattice Boltzmann flexible particle method (LBFPM) in a three-dimensional space at a chord based Reynolds number of 100. The flexibility and wing mass ratios are systematically varied, and their effects on aerodynamic forces and power efficiency are explored. It is demonstrated that large the deformation is controlled by the wing and fluid flow inertia ("added mass) and the wing flexibility. A dynamic balance between the inertia and the appropriate level of the flexibility allows the flexible motion in phase with the driving base and have a much larger rotational velocity or rotational momentum than a rigid wing, resulting in a better performance.

5:50PM E24.00006 Force Element Theory for Finite Wings at Low Reynolds numbers , CHIN-CHOU CHU, JIAN-JHIH LEE, CHENG-TA HSIEH, CHIEN-CHENG CHANG, National Taiwan University — This paper is aimed to examine various contributions to the forces on an impulsively started finite plate from the perspective of a force-element representation. The wing plate has an aspect ratio (AR) between 1 and 3, and is placed at low and high angles of attack (α =5°, 10°, 15°, 30°, 45°, and 60°), while the Reynolds number Re is varied between 100 or 300. The force theory enables us to quantify the contributions to the forces exerted on the plate in terms of all the fluid elements with nonzero vorticity, such as in the tip vortices (TiVs), leading- and trailing-edge vortices (LEV and TEV) as well on the plate surface. The present vorticity force analysis (VFA) was made parallel to the pressure force analysis (PFA) by examining the sectional force contributions along the wing span, but can further extend to include the outer regions (of TiVs). The interplay between the LEV and the TiVs by assessing the relative importance of the transverse as well as the longitudinal vorticity components at various time stages leads to insightful physical explanations of the force mechanisms.

Sunday, November 24, 2013 4:45PM - 5:37PM –

Session E25 Flow Control III - Drag Reduction 320 - Wade Huebsch, West Virginia University

4:45PM E25.00001 Reduction of turbulent skin-friction drag by oscillating discs , DANIEL WISE, PIERRE RICCO, None — A new drag-reduction method, based on the active technique proposed by Ricco & Hahn (2013), i.e. steadily rotating flush-mounted discs, is studied by DNS. The effect of sinusoidally oscillating discs on the turbulent channel-flow drag is investigated at $Re_\tau = 180$, based on the friction velocity of the stationary-wall case and the half channel height. A parametric investigation on the disc diameter, tip velocity and oscillation period yielded a maximum drag reduction of 18.5%. Regions of net power saved, calculated by considering the power spent to enforce the disc motion against the viscous resistance of the fluid, are found to reach up to 6.5% for low disc tip velocities. Significantly, the characteristic time-scale for the oscillating disc forcing is double that for the steadily rotating discs, representing a further step towards industrial implementation. The oscillating disc forcing, similar to the steadily rotating disc forcing, creates streamwise-elongated structures between the discs. These structures - largely unaffected by the periodic wall forcing and persisting throughout the entire period of the oscillation - are the main contributor to the additional Reynolds stresses term created by the disc forcing, and are important for the drag reduction mechanism.

4:58PM E25.00002 Designing smart duct geometries for low frictional losses , GERTRAUD DASCHIEL, Institute of Fluid Mechanics, Karlsruhe Institute of Technology, VERONIKA KRIEGER, JOVAN JOVANOVIC, Institute of Fluid Mechanics, University of Erlangen-Nuremberg, BETTINA FROHNAPFEL, Institute of Fluid Mechanics, Karlsruhe Institute of Technology — In turbulent flows through triangular ducts the friction factor is significantly reduced compared to the well-proven Blasius correlation. The passages of reduced friction are detected close to the duct corners in which the flow also shows a strong tendency in the turbulent fluctuations towards the statistical axisymmetric state. Within the present investigation direct numerical simulations of turbulent flows through non-circular ducts are carried out. The duct shapes are designed with the goal to reduce frictional losses in the turbulent state by forcing turbulent fluctuations towards statistical axisymmetry in a wide part of the flow domain. In this respect, the influence of the corners' opening angle and the surface curvature are investigated. Interestingly, the state of statistical axisymmetry is also reported to lead to a stabilization of disturbances in laminar flows and consequently delay the breakdown to turbulence. From this finding it might be expected that duct geometries leading to this particular statistical properties of the turbulent fluctuations also can have beneficial effects in the delay of the laminar to turbulent transition process. First numerical experiments that tackle this point will be presented.

5:11PM E25.00003 Model-based analysis of the effect of spanwise wall oscillations on drag reduction at high Reynolds numbers , ARMIN ZARE, University of Minnesota, RASHAD MOARREF, Caltech, MIHAILO JOVANOVIC, University of Minnesota — Experiments and numerical simulations have shown that drag-reducing ability of spanwise wall oscillations in turbulent channels deteriorates as the Reynolds number increases. Recent work by Moarref and Jovanovic (J. Fluid Mech., vol. 707, 2012) has demonstrated the predictive power of a model-based approach for controlling turbulent flows. In the present study, we use a linearized stochastically-forced model to reveal the Reynolds number independent effects of wall oscillations on drag reduction. This allows us to extend the predictive capability of our simulation-free approach to high Reynolds numbers. We show that the influence of wall oscillations at low Reynolds numbers is confined to the streamwise and spanwise wavelengths that correspond to the universal inner-scaled eddies in wall turbulence. Since wall oscillations do not suppress large scale eddies, which are responsible for increased drag in the uncontrolled flow, we conclude that wall oscillations have weaker influence on drag reduction at higher Reynolds numbers. In addition, our observations enable predictions of drag reduction trends at high Reynolds numbers.

5:24PM E25.00004 Drag-Reduction Effectiveness of Riblet Films in Adverse Pressure Gradients[1] , AARON BOOMSMA, FOTIS SOTIROPOULOS, University of Minnesota — Riblet films are micro-grooved structures that are widely known to passively reduce skin friction. Past studies have almost solely focused on riblet performance in channel-flows. However, possible applications of riblets include wind turbine blades, gas turbine blades, and other complex bodies that are exposed to non-zero pressure gradient flows—specifically adverse pressure gradients. We use high-resolution large eddy simulations of turbulent flow over three-dimensional riblets under an adverse pressure gradient. We analyze the computed results to quantify drag reduction effectiveness for different riblet shapes and to examine pertinent turbulent structures to gain a fundamental understanding of riblet performance.

[1]Supported by the DOE Wind Energy Consortium

Sunday, November 24, 2013 4:45PM - 5:50PM –

Session E26 Reacting Flows III: Coal & Soot 321 - Babak Shotorban, The University of Alabama in Huntsville

4:45PM E26.00001 Transported PDF modeling for pulverized coal combustion[1] , XINYU ZHAO, DANIEL C. HAWORTH, Pennsylvania State University — A transported probability density function (PDF) method has been applied to the simulation of pulverized coal combustion. A Lagrangian particle/Eulerian mesh algorithm has been employed to treat the gas phase. An independent set of Lagrangian parcels are used to model the solid phase (pulverized coal). The two phases are coupled by particle-source-in-cell technique and by source term redistribution models. Other high-fidelity models, such as photon Monte Carlo radiation model, are integrated into the PDF-coal framework. Temperature and global characteristics of coal combustion are compared with experimental data. Sensitivities of the results to model variations are explored.

[1]Supported by national energy technology laboratory regional university alliance

4:58PM E26.00002 A Study of Turbulence-Chemistry-Soot-Radiation Interaction in Luminous Turbulent Jet Flames[1] , SOMESH ROY, DANIEL HAWORTH, Mechanical and Nuclear Engineering Dept., Pennsylvania State University — A detailed soot model based on method of moments with interpolative closure (MOMIC) is used in RANS simulations of luminous turbulent jet flames using OpenFOAM. A detailed chemical mechanism has been used to describe the chemistry of key soot precursors, and a transported probability density function (tPDF) method has been used to capture the turbulence-chemistry-soot-radiation interactions. The results from the detailed soot model have been compared with those from a semi-empirical, two-equation soot model for accuracy and performance. The effects of turbulence-chemistry-radiation interactions on soot dynamics are isolated and quantified.

[1]This work has been supported by NASA under cooperative agreement NNX07AB40A and by NSF under grant OCI-0904649.

5:11PM E26.00003 Large-eddy simulation of pulverized coal swirl jet flame , MASAYA MUTO, Kyoto University, HIROAKI WATANABE, Central Research Institute of Electric Power Industry, RYOICHI KUROSE, SATORU KOMORI, Kyoto University, SARAVANAN BALUSAMY, SIMONE HOCHGREB, University of Cambridge — Coal is an important energy resource for future demand for electricity, as coal reserves are much more abundant than those of other fossil fuels. In pulverized coal fired power plants, it is very important to improve the technology for the control of environmental pollutants such as nitrogen oxide, sulfur oxide and ash particles including unburned carbon. In order to achieve these requirements, understanding the pulverized coal combustion mechanism is necessary. However, the combustion process of the pulverized coal is not well clarified so far since pulverized coal combustion is a complicated phenomenon in which the maximum flame temperature exceeds 1500 degrees Celsius and some substances which can hardly be measured, for example, radical species and highly reactive solid particles are included. Accordingly, development of new combustion furnaces and burners requires high cost and takes a long period. In this study, a large-eddy simulation (LES) is applied to a pulverized coal combustion field and the results will be compared with the experiment. The results show that present LES can capture the general feature of the pulverized coal swirl jet flame.

5:24PM E26.00004 Damköhler number effects on soot formation and growth in turbulent nonpremixed flames , FABRIZIO BISETTI, ANTONIO ATTILI, King Abdullah University of Science and Technology, MICHAEL E. MUELLER, Princeton University, HEINZ PITSCH, RWTH Aachen University — An analysis of soot formation and growth, based on a set of large simulations of n-heptane/air turbulent nonpremixed combustion, is presented. A detailed chemical mechanism, which includes polycyclic aromatic hydrocarbons, and a high-order method of moments for soot modeling are employed for the first time in the three-dimensional simulation of turbulent sooting flames. The effects of scalar dissipation rate on the soot growth are studied performing three simulations at different Damköhler number while holding the Reynolds number constant. The temperature field is unchanged by the rescaling, due to negligible extinction in all cases. Soot precursors are more sensitive to strain than temperature and their peak concentration decreases by about 40% and 80% as the Damköhler number is reduced by a factor of 2 and 4. It is shown that decreasing the Damköhler number does not affect the soot number density, while the soot mass fraction shows a linear dependence on Damköhler number.

5:37PM E26.00005 Monte Carlo Simulation Of Soot Evolution along Lagrangian Trajectories in a Turbulent Flame , AHMED ABDELGADIR, KUN ZHOU, ANTONIO ATTILI, FABRIZIO BISETTI, Clean Combustion Research Center, KAUST — A newly developed Monte Carlo method is used to simulate soot formation and growth in a turbulent n-heptane/air flame. The Monte Carlo method is used to simulate the soot evolution along selected Lagragnian trajectories obtained from a direct numerical simulation of a turbulent sooting jet flame [Attili *et al.*, Direct and Large-Eddy Simulation 9, Springer, 2013] based on a high-order method of moments. The method adopts an operator splitting approach, which splits the deterministic processes (nucleation, surface growth and oxidation) from coagulation, which is treated stochastically. The purpose of this work is to assess the solution based on the moment method and to investigate the soot particle size distribution (PSD) that is not available in methods of moments. Nucleation and coagulation have the greatest effect on the PSD, therefore, various coagulation models are considered. Along each trajectory, one or more rapid nucleation events occur, affecting the shape of the PSD. It is shown that oxidation and surface growth affect the PSD quantitatively, but do not change the shape significantly.

Sunday, November 24, 2013 4:45PM - 5:50PM –

Session E27 Supersonic and Hypersonic Flows: Shock Capturing and Focusing Spirit of Pittsburgh Ballroom A - Veronica Eliasson, University of Southern California

4:45PM E27.00001 Shock Wave Mitigation Using Lessons Learned from Shock Focusing Techniques , QIAN WAN, VERONICA ELIASSON, University of Southern California AME Department — Shock wave mitigation in channels has been a topic of much attention in the shock wave community. One approach is to use obstacles of various geometries arranged in different patterns to attenuate an incident shock wave. Following the numerical work of A. Chaudhuri et al. (2012), which used cylinders, squares and triangles placed in staggered and non-staggered subsequent columns, we present simulations using a different obstacle pattern that more efficiently attenuates shock waves. Instead of using a matrix of obstacles, we have investigated square-shaped obstacles placed along a logarithmic spiral curve inspired by our previous work on shock focusing using logarithmic spirals. Results indicate that a logarithmic spiral could be an efficient way to collect and reflect the main part of the incident shock wave, thus improving techniques of shock wave attenuation in channels.

4:58PM E27.00002 Shock focusing using multiple micro-blast waves , SHI QIU, ZIJIE ZHANG, VERONICA ELIASSON, University of Southern California AME department — Numerical simulations have been used to study shock focusing effects from multiple micro-blast waves and to determine shock front amplification mechanisms as the wave converges. Overture, a partial differential equation solver, was used to solve the Euler equations with a second-order accurate Godunov algorithm. Adaptive mesh refinement was used to improve the accuracy and reduce the computational time. The early stage of each micro-blast wave was initialized using Taylor's similarity laws. The total energy of the blast waves was kept constant, but the number of blast waves and their respective size were varied from case to case. Results show that through careful geometrical arrangement and timing of the initialization of the charge, multiple micro-blast waves can be combined to yield more extreme thermodynamic conditions at the focal area than compared to a large blast wave with the same total initial energy.

5:11PM E27.00003 Experiments and simulations of shock focusing in thin water-filled convergent structures[1] , VERONICA ELIASSON, CHUANXI WANG, SHI QIU, University of Southern California — A shock wave generated by an underwater explosion will impact any nearby surrounding structure and the fluid-structure interaction during the shock impact has to be understood to keep the structure safe from damage. Extremely high pressures will be generated, and the time of impact is on the order of a few milliseconds. In this work, the fluid-structure interaction during shock wave impact on water-filled convergent structures is studied extensively to assess the strength and dynamical response of the surrounding structure to lead to viable future design considerations to minimize or avoid damage. The response of light-weight composite materials are compared to that of steel structures. Experiments using high-speed non-invasive schlieren techniques and finite element numerical simulations have been performed. Results show that the fluid-structure interaction during shock impact is highly dependent on the thickness and material properties of the surrounding convergent structure. Precursor waves in the water ahead of the incident shock wave behave differently from case to case, and will be addressed in this presentation.

[1]Funded by ONR Solid Mechanics Program (N00014-06-1-073)

5:24PM E27.00004 Effects of Heat Conduction on Artificial Viscosity Methods for Shock Capturing[1] , ANDREW COOK, LLNL — We investigate the efficacy of artificial thermal conductivity for shock capturing. The conductivity model is derived from artificial bulk and shear viscosities, such that stagnation enthalpy remains constant across shocks. By thus fixing the Prandtl number, more physical shock profiles are obtained, only on a larger scale. The conductivity model does not contain any empirical constants. It increases the net dissipation of a computational algorithm but is found to better preserve symmetry and produce more robust solutions for strong-shock problems.

[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.

5:37PM E27.00005 Numerical simulations of blast/shock wave propagations after nuclear explosions[1] , SEUNGHO SONG, Dept. ME, Yonsei University, JUNG-IL CHOI, YIBAO LI, CHANGHOON LEE, Dept. CSE, Yonsei University — Pressure waves develop immediately after nuclear explosions and start to move outward from the fireball. The most of initial damages are caused by the blast waves. We performed the blast wave propagations by solving two-dimensional and axisymmetric Euler equations. For shock capturing, inviscid fluxes are discretized using a variant of the piecewise parabolic method (PPM) and an approximate Riemann solver based on Roe's method is used. A clean air burst of fireball above the ground zero is considered. The initial condition of fireball is given at the point of breakaway that shock waves are appeared on the surface of the fireball. The growth of fireball is also calculated by solving one-dimensional radiation hydrodynamics (RHD) equation from point explosion. Characteristics of the blast wave propagations due to the various heights of burst and amount of the nuclear detonations are investigated. The results of parametric studies will be shown in the final presentation.

[1]Supported by Agency for Defense Development.

Sunday, November 24, 2013 4:45PM - 5:37PM –

Session E28 Viscous Flows I: Flow Past Interferences Spirit of Pittsburgh Ballroom B/C - Richard McLaughlin, The University of North Carolina at Chapel Hill

4:45PM E28.00001 Oscillatory Stokes Flow Past a Slip Cylinder[1] , D. PALANIAPPAN, Texas A&M University - Corpus Christi — Two-dimensional transient slow viscous flow past a circular cylinder with Navier slip boundary conditions is considered in the limit of low-Reynolds number. The oscillatory Stokes flow problem around a cylinder is solved using the stream function method leading to an analytic solution in terms of modified Bessel functions of the second kind. The corresponding steady-state behavior yields the familiar paradoxical result first detected by Stokes. It is noted that the two key parameters, viz., the frequency λ, and the slip coefficient ξ have a significant impact on the flow field in the vicinity of the cylinder contour. In the limit of very low frequency, the flow is dominated by a term containing a well-known biharmonic function found by Stokes that has a singular behavior at infinity. Local streamlines for small times show interesting flow patterns. Attached eddies due to flow separation - observed in the no-slip case - either get detached or pushed away from the cylinder surface as ξ is varied. Computed asymptotic results predict that the flow exhibits inviscid behavior far away from the cylinder in the frequency range $0 < \lambda \ll 1$. Although the frequency of oscillations is finite, our exact solutions reveal fairly rapid transitions in the flow domain.

[1]Research Enhancement grant, TAMUCC

4:58PM E28.00002 Viscous power-law flow past a finite flat plate , LING XU, Georgia State University, MONIKA NITSCHE, University of New Mexico — Viscous flow past a finite flat plate is studied numerically, using a high order implicit finite difference scheme. The plate moves in direction normal to itself with velocity $V_\infty = t^p$. We present the dependence of the vorticity evolution, streamlines and streaklines on $p \in [0, 2]$ and on Reynolds number $Re \in [250, 2000]$, and compare with experimental results of Pullin & Perry (1980). We observe that, unlike in the p=0 case, for $p \neq 0$ the vortex core position oscillates as it moves away from the plate.

5:11PM E28.00003 Experimental and modeling study of global circulation by bent rod precession in low Reynolds number flows[1] , ROBERTO CAMASSA, J.D. MARTINDALE, RICHARD MCLAUGHLIN, Department of Mathematics, University of North Carolina at Chapel Hill, LEANDRA VICCI, Department of Computer Science, University of North Carolina at Chapel Hill, LONGHUA ZHAO, Department of Mathematics, Case Western Reserve University, UNC JOINT FLUIDS LAB TEAM — The precessing motion of a bent rod over a plane in viscous dominated regimes can generate global fluid flow structures in the form of recirculating tori. Such motion can play an important role in the development of multicellular organisms, where primary cilia are the main agent for the embryonic forms of nutrient circulation. Results from an experimental investigation using PIV techniques to analyze the flow field will be presented and compared with a first principle theory based on slender body approximations. While good qualitative agreement can be achieved with Blake images enforcing the no-slip condition at the plane, quantitative agreement requires a more sophisticated approach, which will be outlined.

[1]We acknowledge funding received from the following NSF grants: RTG DMS-0943851 and DMS-1009750.

5:24PM E28.00004 The Oscillatory Motion of a Sphere in a Stokes Flow , FINN BOX, School of Physics, University of Manchester, ALICE THOMPSON, School of Mathematics, University of Manchester, TOM MULLIN, School of Physics, University of Manchester — We report results of an experimental investigation into the dynamic response of a single sphere to magnetic forcing and the resultant motion of the surrounding viscous fluid. Permanent magnets embedded into the surface of a neutrally buoyant sphere enable actuation of torsional oscillations of the sphere through the application of an alternating magnetic field. The applied field induces a torque on the embedded magnets, and the torsional response of the sphere to magnetic forcing has been systematically characterized as a function of the dimensionless forcing parameter $F=8\pi\mu\ a^3\omega$. Excellent agreement is found between the experimentally observed and numerically computed behavior of the sphere. Furthermore, the flow generated by the rotary motion of a sphere has visualized using Particle Image Velocimetry and good agreement is also found between the observed and the analytic solution for the fluid velocity as a function of radial distance.

Sunday, November 24, 2013 4:45PM - 5:50PM –

Session E30 Turbulence: Shear Layers I - Simulations 408 - Cyrus Madnia, University at Buffalo - The State University of New York

4:45PM E30.00001 Characteristics of the Turbulent/Non-Turbulent Interface in Compressible Shear Layers , NAVID S. VAGHEFI, CYRUS K. MADNIA, SUNY at Buffalo — Direct numerical simulation (DNS) of temporally evolving compressible turbulent shear layer at varying convective Mach numbers are used to assess the dynamics of the flow across the turbulent/non-turbulent (T/NT) interface separating the turbulent and the irrotational regions. This interface is detected by using a certain threshold for the vorticity norm. The conditional flow statistics based on the normal distance from the T/NT interface plane are compared for different convective Mach numbers. It is shown that the T/NT interface thickness for compressible cases examined is of order of the Taylor microscale, similar to the previous studies for incompressible flows with mean shear. Various terms in kinetic energy and vorticity transport equations are examined in order to determine the effects of compressibility on the transport mechanisms across the T/NT interface.

4:58PM E30.00002 Numerical study of shear layer instability in transverse jets[1] , KRISHNAN MAHESH, PRAHLADH IYER, University of Minnesota — Direct numerical simulations are performed to study the transition from absolute to convective instability for incompressible jets in crossflow using an unstructured finite volume solver. Flow conditions are based on experimental conditions of Megerian $et\ al.$ (2007) for a flush injected jet. Results obtained from simulated jet velocity ratios of 2 and 4 are compared with experimental data (Getsinger $et\ al.$ 2011 and Megerian $et\ al.$ 2007) and show good agreement in instantaneous and time averaged flow characteristics as well as velocity spectra. Proper orthogonal decomposition (POD) and Koopman mode decomposition of the three-dimensional flow field is performed to identify the dominant flow features and their corresponding frequencies.

[1]Supported by AFOSR

5:11PM E30.00003 The Turbulent/Non-Turbulent Interface in Non-Premixed Reacting Mixing Layers , REZA JAHANBAKHSHI, NAVID S. VAGHEFI, CYRUS K. MADNIA, SUNY at Buffalo — The results of the direct numerical simulation (DNS) of temporally evolving reacting mixing layer are used to study the flow characteristics across the turbulent/non-turbulent (T/NT) interface separating the turbulent and the irrotational regions. This interface is detected by using a certain threshold for the vorticity norm. The compressible form of the conservation equations for mass, momentum, and energy are solved. The hydrogen-air combustion is mimicked by a one-step global reaction. The infinitely fast chemistry approximation is used. The dynamic viscosity and heat capacities depend on local temperature and species mass fractions. The main objective of this work is to study the effects of heat release and flame location on kinematics and dynamics of the T/NT interface.

5:24PM E30.00004 Direct Numerical Simulation of a Temporal Mixing Layer and Detection of the Turbulent/Non-Turbulent Interface , FABIAN HENNIG, JONAS BOSCHUNG, MICHAEL GAUDING, NORBERT PETERS, RWTH Aachen University — The direct numerical simulation of a temporally evolving mixing layer is presented. Using the DNS data we compare two different approaches of detecting the so called turbulent/non-turbulent interface that is found between the fully turbulent and the irrotational outer flow. Standard and conditional statistics are evaluated and compared with literature results.

5:37PM E30.00005 Temporal behavior of strong shear layers in high Reynolds number turbulence , PRADEEP K. JHA, TAKASHI ISHIHARA, Nagoya University, JST CREST — High resolution direct numerical simulation (DNS) of isotropic turbulence with the Taylor micro-scale Reynolds number $R_\lambda = O(10^3)$ on 4096^3 grid points was used to study the temporal behavior of strong shear layers in high Reynolds number turbulence. A time span of $10\tau_\eta = 2.55\lambda/u'$ was simulated and analyzed, where τ_η is the Kolmogorov time-scale, λ is the Taylor micro-scale and u' is the rms value of the velocity fluctuations. Detailed visualization showed excellent correspondence between regions with high enstrophy values and the existence of strong shear layers. Reasonably close-packed elongated strong vortices were found to exist in layer-like regions with thickness of the order of λ. A quantitative analysis of the DNS data showed that in these strong shear layers, strong vortices interact with the neighboring vortices and move drastically at a speed of the order of u', maintaining an effectively constant distance between each other. The average size of these peak vortices also remains quasi-time-independent. The strong shear layers at the interfacial region remain sharp during the time evolution. These shear layers are significant intermittent structures of high Reynolds number turbulence.

Sunday, November 24, 2013 4:45PM - 5:37PM –

Session E31 Porous Media Flows IV: Electrochemical and Heat Transfer Devices 402 - Ying Sun, Drexel University

4:45PM E31.00001 Capacitive Charging and Desalination with Porous Electrodes , HOWARD HU, MENGYING LI, HAIM BAU, University of Pennsylvania — Electrochemical capacitors (ECs) are attractive storage devices with many advantages over traditional batteries. In contrast to batteries, ECs store energy in the electric double layer and do not undergo electrochemical reactions during charging and discharging. In this work, we examine the charging/discharging process of an EC cell consisting of a conductive, granular medium confined between two parallel, current collecting, electrodes. The granular particles are porous, assembled by aggregation, and saturated with an electrolyte solution. They are separated into two compartments with an ion-permeable, electrically insulating membrane. The Debye screening length around and within porous particles is small compared to the characteristic size of the pores. We will present a mathematical model based on Poisson-Nernst-Planck equations to describe the charging/discharging process in the EC cell. Using this model, the ion distribution and potential variation within the cell are solved numerically as functions of time when the current collecting electrodes are subjected to a step change and to time-periodic alternations in electrodes' potentials. The same model can be also used for potential desalination applications.

4:58PM E31.00002 GPU-enabled Computational Model of Electrochemical Energy Storage Systems , CHARLES ANDERSEN, GANG QIU, NAGARAJAN KANDASAMY, YING SUN, Drexel University — We present a computational model of a Redox Flow Battery (RFB), which uses real pore-scale fiber geometry obtained through X-ray computed tomography (XCT). Our pore-scale approach is in contrast to the more common volume-averaged model, which considers the domain as a homogenous medium of uniform porosity. We apply a finite volume method to solve the coupled species and charge transport equations. The flow field in our system is evaluated using the Lattice Boltzmann method (LBM). To resolve the governing equations at the pore-scale of carbon fibers, which are on the order of tens of microns, is a highly computationally expensive task. To overcome this challenge, in lieu of traditional implementation with Message Passing Interface (MPI), we employ the use of Graphics Processing Units (GPUs) as a means of parallelization. The Butler-Volmer equation provides a coupling between the species and charge equations on the fiber surface. Scalability of the GPU implementation is examined along with the effects of fiber geometry, porosity, and flow rate on battery performance.

5:11PM E31.00003 Simulation of water splitting reaction in porous media using Random Walk particle tracking method , NIMA RAHMATIAN, University of Florida, JÖRG PETRASCH, Vorarlberg University of Applied Sciences, RENWEI MEI, JAMES KLAUSNER, University of Florida — Water splitting using iron-based looping process is a well-known method to produce high purity hydrogen. A stable porous structure is best suited for the reaction over many cycles due to high surface area. In order to simulate the reacting flow in the porous structure Random Walk method is used due to its ability to handle stiff reaction kinetics and varying hydrodynamic dispersion tensor caused by pore-level velocity fluctuations. Because of significant variation in bulk density during conversion of steam to hydrogen, Random Walk formulation needs to be modified to account for bulk density variations and source term due to chemical reaction. The species transport equation is recast in the form of Fokker-Planck equation and the trajectories of fluid particles are obtained by solving an appropriate Langevin equation that has additional drift terms due to spatial variations in bulk density and dispersion tensor. The source term is accounted for by changing the number or the composition of fluid particles based on the reaction kinetics. The treatment for each new term is validated using highly resolved finite difference solution. A bench-scale reactor for hydrogen production is simulated and excellent agreement with the measured hydrogen production rate is obtained.

5:24PM E31.00004 Estimation of Porous Medium Tortuosity Directly from Flow Path Lines , SURYANARAYANA PAKALAPATI, ISMAIL CELIK, West Virginia University — A thorough understanding of transport processes inside porous materials is vital for improving the efficiency of energy devices such as fuel cells and batteries. Continuum simulations of porous media make use of parameters such as porosity and tortuosity to account for the influence of the actual pore geometry and orientation on the transport processes. In most studies the tortuosity is treated as an adjustable parameter which is calibrated to match the predictions with the experiments. In this study a direct method is utilized to estimate the tortuosity of a porous medium. The actual geometry of a fuel cell electrode is obtained from an experimental study where the porous structure is reconstructed from slice images. The detailed geometry of porous medium is used to simulate fully resolved fluid flow through the pores. Stream lines are then generated which show the actual paths taken by the fluid flowing through the porous medium. The lengths of these path lines are then used to calculate the tortuosity of the porous medium by employing the actual definition of the tortuosity. It is shown that the tortuosities obtained in this way are smaller than the typical values reported in literature.

Sunday, November 24, 2013 4:45PM - 5:50PM –

Session E32 Surface Tension Effects III: General Interfacial Phenomena 403 - Sushanta Mitra, University of Alberta

4:45PM E32.00001 Low Interfacial Tension Measurement with Synthetic Schlieren Imaging , AVANISH MISHRA, VARUN KULKARNI, JIAN-WEI KHOR, STEVE WERELEY, Purdue University — Interfacial tension in liquid-liquid systems can be deduced by measuring shape of the static meniscus profile near a vertical solid wall. When interfacial tension is low (less than 1 mN/m), meniscus becomes too small in size to be properly measured by side view imaging. In this work, instead of relying on the side view imaging, we use the Free Surface Synthetic Schlieren (FS-SS) method for the measurement of surface gradient of the meniscus from top view images. The interfacial tension is estimated by fitting the measured surface gradient profile with the theoretical results. Since this method utilizes distortion in a background pattern for estimating the surface gradient, it allows us to measure very low values of the interfacial tension.

4:58PM E32.00002 Inertial Rise in Short Capillaries , OREST SHARDT, PRASHANT WAGHMARE, SUSHANTA MITRA, JOS DERKSEN, University of Alberta — We investigate the primarily inertial rise of liquid in vertical glass capillaries that are shorter than the equilibrium rise height (Jurin height). We focus on the behavior of the liquid upon reaching the top of the capillary and use high-speed imaging to observe the motion of the liquid-air interface with high spatial and temporal resolution. We examine the dependence of the interface behavior on the meniscus speed and capillary height and describe a new phenomenon. Upon reaching the upper edge of a sufficiently short capillary, the meniscus inverts, rises upward, and bulges out radially. The bulging liquid then wets the external surface of the capillary and slides down. The meniscus inside the capillary retracts, falling below the upper edge, and then oscillates vertically with decaying amplitude, inverting several times before reaching a steady shape. A theoretical analysis is used to interpret the conditions required for this phenomenon to occur. A key assumption in the analysis is that the transient flow is inertial and therefore the capillary driving force is balanced by the weight and inertia of the rising liquid column while viscous forces are comparatively small. The analysis points to the possibility of obtaining previously-unseen behavior under reduced gravity.

5:11PM E32.00003 Controlled Coating of Self-Assembled Sphere Clusters by Gravitational Forcing , STEVEN G. JONES, ABHINAV AHUJA, VIVIAN TRUONG, SCOTT S.H. TSAI, Department of Mechanical and Industrial Engineering, Ryerson University — The motion of a spherical particle moving through a liquid-liquid interface due to gravitational force is one of the classical problems of fluid dynamics. In some situations, a single particle has insufficient gravitational energy to break through the interface, but a cluster of multiple particles overcomes the interfacial tension energy barrier to pass through. Here we show with experiments that particles self-assemble into clusters upon settling at an oil-water interface. When a cluster is sufficiently large, the cluster will pass through the interface and becomes conformally coated. We find that the number of particles inside one cluster is proportional to a power-law of the Bond number, which describes the ratio between gravitational and surface tension energies. We demonstrate that the size of the coated particle cluster can be tuned by altering the radius of the particles and changing the interfacial tension.

5:24PM E32.00004 Electric field driven bubble motion in microgravity[1] , BORIS KHUSID, DANA QASEM, EZINWA ELELE, JOHN TANG, YUEYANG SHEN, New Jersey Institute of Technology, Newark, NJ — The lack of the gravity-driven gas-liquid phase separation in microgravity has severely compromised a wide range of space technologies. The proposed electro-hydrodynamic (EHD) control of the bubble motion in microgravity employs an electric force generated by an alternating current (AC) field applied directly to a fluid via capacitive coupling to external electrodes. Contrary to the currently available direct current (DC) field-based microgravity techniques, the EHD method employs flow- and field-induced forces to drive bubbles and suppresses electro-chemical reactions at the fluid/electrode interface. The overall goal of the parabolic flight tests planned in July-Aug 2013 is the validation of the EHD method for the control and manipulation of bubbles in microgravity. We will present test results at the meeting.

[1]The work is supported by NASA grant NNX09AK06G.

5:37PM E32.00005 Effects of particle self-assembly and structural disjoining pressure on wetting kinetics of nanofluid droplet[1] , GUI LU, Dept. of Thermal Engineering, Tsinghua Universtiy, HAN HU, Dept. of Mechanical Engineering and Mechanics, Drexel University, YUANYUAN DUAN, Dept. of Thermal Engineering, Tsinghua Universtiy, YING SUN, Dept. of Mechanical Engineering and Mechanics, Drexel University — The wettability of nanofluids, fluids containing suspensions of nanometer-sized particles, is of particular interest to microfluidic systems. Previous studies showed that the self-assembly of nanoparticles in the vicinity of the contact line gives rise to a structural disjoining pressure, which greatly affects the wettability of nanofluid droplets of micron size or larger. In this study, dynamic wetting of water nano-droplets containing non-surfactant gold nanoparticles on a gold substrate was studied via molecular dynamics simulations to examine the effects nanoparticle self-assembly. To mimic the effect of structural disjoining pressure, the excess disjoining pressure was calculated for a pure water film on a gold substrate with a smooth surface on one end and ordered nano-pillar structures on the other. The results show that the addition of non-surfactant nanoparticles hinders the nano-second droplet wetting process, attributed to the increases in both surface tension of the nanofluid and friction between nanofluid and substrate. The spreading enhancement of nanofluids due to the presence of structural disjoining pressure as a result of nanoparticle ordering is not the case for this nano-droplet spreading system.

[1]NSFC (No. 21176133) and the US National Science Foundation (Grant No. CAREER-0968927 and No. DMR-1104835).

Sunday, November 24, 2013 4:45PM - 5:50PM –

Session E33 Drops V: Buoyancy-Driven Motion 404 - Thomas Ward, Iowa State University

4:45PM E33.00001 Sedimentation and deformation of an aqueous sodium hydroxide drop in vegetable oil[1] , ANDREW WHITE, HYAQUINO HYACINTHE, THOMAS WARD, Iowa State University — The addition of water droplets in fuels is known to provide benefits such as decreased Nitrous Oxide NOx emissions. Unfortunately the shelf life of a water-fuel emulsion is limited by the sedimentation rate of the water droplets. It is well known that adding surfactants can significantly slow the sedimentation rate due to the introduction of Marangoni stresses. In the case of a vegetable oil fuel, adding sodium hydroxide ($NaOH$) to the water droplets will produce surfactants through saponification in the form of sodium-carboxylate salts. Pendant drops of aqueous $NaOH$ solutions with pH between 11 and 13 will be suspended in several oils such as corn, olive, canola and soybean oil in order to measure the interfacial tension. The change in interfacial tension with time will be used to estimate the surfactant concentration and the saponification rate. Then individual drops will be placed in the oils to observe the settling velocity and drop deformation.

[1]NSF CBET

4:58PM E33.00002 Steady, axisymmetric, buoyancy-driven motion of a drop rising through a less viscous liquid , LOUIS J. STEYTLER, University of Illinois at Urbana-Champaign, JANES Q. FENG, Optomec, Inc., ARNE J. PEARLSTEIN, University of Illinois at Urbana-Champaign — We report finite-element computations of steady, axisymmetric motion of a drop rising through a much less viscous liquid. The results focus on drops of liquid CO_2 rising through water, under conditions where the hydrostatic pressure variation is small enough for the density and viscosity variation in the drop to be neglected. A range of novel drop shapes is reported, with a recirculating flow region sometimes developing near the front or near the centerline of the drop. We also compute and discuss several measures of the velocity and drop shape relevant to transport of a passive scalar from the drop to the suspending liquid, or vice versa.

5:11PM E33.00003 Gravitational Interactions of Two Small Evaporating Drops , MICHAEL ROTHER, University of Minnesota Duluth — Relative trajectories are calculated for two sedimenting spherical drops with exact methods for determining the hydrodynamic forces at finite Stokes number and low Reynolds number. The drops are losing mass by isothermal evaporation controlled by diffusion, and bispherical coordinates are used to solve for the vapor concentration between the two liquid spheres. 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. The effect of evaporation is studied by comparison with trajectories for two drops of constant mass. The effect of the second drop on mass loss is analyzed by comparing trajectory results with those for two interacting drops, each evaporating at the isolated drop rate. An important application is to raindrop growth. For water droplets in the atmosphere, at drop radii between 10 and 30 μm, drop inertia is important while the Reynolds based on the surrounding air is still small.

5:24PM E33.00004 Ascending dynamics of a swarm of drops in a stratified fluids[1] , AREZOO ARDEKANI, MORTEZA BAYAREH, SADEGH DABIRI, University of Notre Dame — The motion of drops and bubbles in stratified fluids has several natural and industrial applications such as bubbles rising across pycnoclines in marine and aquatic environments, oil spills, ocean sequestration of CO2, and bubble mixers used for lake/reservoir destratification and aeration. Even though the motion of bubbles and drops in a homogenous fluid has been extensively studied, their motion in a stratified fluid has not been explored. We perform direct numerical simulation of three-dimensional motion of a swarm of drops rising in a linearly stratified fluid. The pair correlation function is calculated to characterize the microstructure formation. The results show stronger tendency to form horizontal clusters of drops in a stratified fluid compared to a homogenous fluid. The mean rise velocity of the swarm of drops is reduced compared to the one in a homogenous fluid.

[1]This work is supported by National Science Foundation Grant CBET-1066545.

5:37PM E33.00005 Low Temperature Distillation for Desalination , WILLIAM SCHULTZ, University of Michigan — We examine a unique configuration that combines the evaporator and condenser in a low temperature distillation process. The low temperature (pressure) container is designed to use waste heat from a power plant as the hot source and a water reservoir as the cold source. Fresh and saline streams of droplets in close proximity create interesting hydrodynamic challenges for the directional stability of the droplets.

Sunday, November 24, 2013 4:45PM - 5:50PM –

Session E34 Drops VI: Bouncing Drops 405 - David Hu, Georgia Institute of Technology

4:45PM E34.00001 Droplets walking in a rotating frame: from quantized orbits to wavelike statistics[1] , DANIEL M. HARRIS, JOHN W.M. BUSH, Massachusetts Institute of Technology — We present the results of an experimental investigation of a droplet walking on the surface of a vibrating rotating fluid bath. Particular attention is given to demonstrating that the stable quantized orbits reported by Fort et al. (PNAS, 2010) arise only for a finite range of vibrational forcing, above which chaotic trajectories with wavelike statistics arise. We first present a detailed characterization of the emergence of orbital quantization, and then examine the system behavior at higher driving amplitudes. As the vibrational forcing is increased progressively, stable circular orbits are succeeded by wobbling orbits with, in turn, stationary and drifting orbital centers. Subsequently, there is a transition to wobble-and-leap dynamics, in which wobbling of increasing amplitude about a stationary center is punctuated by the orbital center leaping approximately half a Faraday wavelength. Finally, in the limit of high vibrational forcing, irregular chaotic trajectories emerge, characterized by a wavelike statistical behavior that reflects the persistent dynamic influence of the unstable orbital states.

[1]The authors gratefully acknowledge the financial support of the NSF through Grant CBET-0966452, and DMH through the Graduate Research Fellowship Program.

4:58PM E34.00002 Pilot-wave dynamics in a rotating frame: on the emergence of orbital quantization[1] , ANAND OZA, DANIEL HARRIS, RODOLFO ROSALES, JOHN BUSH, Massachusetts Institute of Technology — We present the results of a theoretical investigation of droplets walking on a rotating vibrating fluid bath. The droplet's trajectory is described in terms of an integro-differential equation that incorporates the influence of its propulsive wave force. Predictions for the dependence of the orbital radius on the bath's rotation rate compare favorably with experimental data and capture the progression from continuous to quantized orbits as the vibrational acceleration is increased. The orbital quantization is rationalized by assessing the stability of the orbital solutions, and may be understood as resulting directly from the dynamic constraint imposed on the drop by its monochromatic guiding wave. The stability analysis also predicts the existence of wobbling orbital states reported in recent experiments, and the virtual absence of stable orbits in the limit of large vibrational forcing.

[1]The authors acknowledge the generous financial support of the NSF through Grant CBET-0966452.

5:11PM E34.00003 Quantization of a particle guided by its own pilot-wave , STÉPHANE PERRARD, Laboratoire MSC, Université Paris Diderot, UMR 7057, MATTHIEU LABOUSSE, EMMANUEL FORT, Institut Langevin, Université Paris Diderot, YVES COUDER, Laboratoire MSC, Université Paris Diderot, UMR 7057 — The association of a particle and a wave can be obtained even at a macroscopic scale, using a simple experimental set up. A liquid bath is set into vertical oscillation so that any drop deposit on it has never the time to break the air layer under it. The drop is always ejected from the bath by the vibration and can then live for hours. The impact generates waves at the surface of the bath which can propel the drop. It becomes a walker, the self-propelled entity formed by a bouncing droplet and its associated surface wave. This system has already shown surprising wave-particle duality as single particle diffraction or Bohr-Sommerfeld quantization of level when the drop is submitted to a transverse force. We now study its motion when the walker is submitted to a central force, which can be tuned at will. In particular, I will present our results in the case of a 2D harmonic potential well. In the case of a strong coupling between the waves and the bouncing drop, the walker exhibit a discrete set of state where the angular momentum and the spatial extend of each level are quantized. Disordered trajectories also appear, as an intermittency between the pure eigenmodes through transitions between them.

5:24PM E34.00004 "Quantum" interference with bouncing drops , TOMAS BOHR, ANDERS ANDERSEN, JACOB MADSEN, CHRISTIAN REICHELT, Physics Department, The Technical University of Denmark, BENNY LAUTRUP, CLIVE ELLEGAARD, MOGENS LEVINSEN, The Niels Bohr Institute, University of Copenhagen — In a series of recent papers (most recently D. Harris, J. Moukhtar, E. Fort, Y. Couder and J. Bush, Phys. Rev. E **88**, 011001(R) (2013)) Yves Couder and collaborators have explored the dynamics of walking drops on the surface of a vibrated bath of silicon oil and have demonstrated a close analogy to quantum phenomena. The bouncing drop together with the surface wave that it excites seems to be very similar to the pilot wave envisaged by de Broglie for quantum particles. In particular, Couder and Fort (Phys. Rev. Lett. **97**, 154101 (2006)) have studied a double slit experiment with walking drops, where an interference pattern identical to the quantum version is found even though it is possible to follow the orbits of the drops and unambigously determine which slit it goes through, something which in quantum mechanics would be ruled out by the Heisenberg uncertainly relations. We have repeated the experiment and present a somewhat more complicated picture. Theoretically, we study a Schrödinger equation with a source term originating from a localised "particle" being simultaneously guided by the wave. We present simple solutions to such a field theory and discuss the fundamental difficulties met by such a theory in order to comply with quantum mechanics.

5:37PM E34.00005 The hydrodynamic boost factor for walking droplets[1] , JOHN BUSH, ANAND OZA, JAN MOLACEK, MIT — It has recently been demonstrated that droplets walking on a vibrating fluid bath exhibit several dynamical features previously thought to be peculiar to the microscopic realm. Such "walkers," consisting of droplet plus guiding wave field, are spatially extended objects. We here examine the dependence of the walker mass and momentum on its velocity. Doing so indicates that in certain parameter regimes, the walker dynamics may be described in terms of the inviscid mechanics of a particle with a speed-dependent mass. Drawing an analogy with relativistic mechanics, we define a hydrodynamic boost factor for these walking droplets, consideration of which provides rationale for the anomalous radii of walkers executing circular orbits.

[1]We gratefully acknowledge the support of the NSF through CBET-0966452.

Sunday, November 24, 2013 4:45PM - 5:50PM –

Session E35 Suspensions III: Confined Flows 406 - Jeffrey F. Morris, City College of New York

4:45PM E35.00001 Shear stress developed on concentrated suspensions of large particles in turbulent shear flow , ESPERANZA LINARES GUERRERO, MELANY HUNT, Caltech — Experiments were performed on concentrated suspensions of relatively large (mm size) non-spherical particles in an aqueous glycerine mixture. The suspension was sheared using the same coaxial-cylinder rheometer used by Koos *et al.*(2012) in which the outer cylinder rotated while the inner one was fixed. The rheometer walls were roughened to avoid slip. Torque measurements for pure fluid and no particles were performed to check for the presence of turbulence. For low Reynolds number ($0.3 - 3 \times 10^3$), the torque measurements compare favorably with the theoretical results for Couette flow but for higher Reynolds ($4 \times 10^3 - 1 \times 10^5$), the torques measured are higher than the ones predicted for a laminar flow. Torque measurements of suspensions of varying concentrations of polystyrene particles were performed. Neutrally and non-neutrally buoyant configurations were studied. To account for particle migration and obtain the local solid fraction, visualization of the flow at the inner wall was performed. Results of the effect of particles in a turbulent shear flow will be presented where focus will be given to distinguish whether the flow is dominated by particle interactions or hydrodynamic forces and the influence that the solid fraction has on these mechanisms.

4:58PM E35.00002 Modeling the Behavior of Confined Colloidal Particles Under Shear Flow , FRANCES MACKAY, COLIN DENNISTON, University of Western Ontario, MIKKO KARTTUNEN, University of Waterloo — Numerically, we investigate the behavior of systems of colloidal particles confined between two parallel walls under steady shear flow. We model these particles using molecular dynamics techniques, with hydrodynamic interactions implemented through the use of a lattice-Boltzmann fluid. Starting from an initially ordered particle arrangement, the system evolves into a variety of configurations depending on the volume fraction and shear rate used. The particles either reorder into hexagonally ordered layers aligned along the flow, form purely disordered layers, or separate into higher volume fraction, ordered layers near the walls, and lower volume fraction, disordered middle layers. We present results in the form of a phase diagram, showing the per-layer behavior as a function of volume fraction and shear rate. In addition, by tracking the positions of individual particles we show that the onset and persistence of disorder in these systems is characterized by an exchange of particles between adjacent layers.

5:11PM E35.00003 Suspension microstructure in a microporous flow , THARANGA PERERA, JAMES GILCHRIST, Lehigh University, LEHIGH UNIVERSITY COLLABORATION — Suspension flows in porous networks are common in applications such as filtration, oil and extraction processes and in biological systems. Knowing the microstructure in such systems can shed light to the understanding of the underlying physical mechanisms as well as the rheological behavior in such flows. An experimental technique based on Confocal laser scanning microscopy was utilized to determine the microstructure of near-hard-sphere microparticle suspensions based on experimentally obtained 3D particle positions. Previous investigations on suspension structure show results of simulation-like quality produced by this technique. This work outlines how the microstructure of a colloidal suspension evolves as it flows around a post in a well-structured micro-porous medium. The formation of strings under shear closer the post as well as ordering due to wall effects are noticed.

5:24PM E35.00004 Microfluidic flow-stabilized solids: formation and deformation , CARLOS ORTIZ, ROBERT RIEHN, KAREN DANIELS, North Carolina State University — The spectrum of non-affine cooperative rearrangements of amorphous materials is central to understand its time-evolution, mechanical properties, and dynamic susceptibility. We report experiments on the structure and elastic properties of a flow-stabilized amorphous solid that is subject to thermal fluctuations. We study flow-stabilized (Pe≈2-20) quasi-2D heaps composed of a bidisperse mixture of sterically-stabilized submicron polystyrene microspheres. Using a microfluidic device, we control the fluid stress applied on the quasi-static heap, allowing us to deform heaps to maximum strains of up to 10% and track the real-time propagation of the local deformation. We then reverse the applied stress perturbation to reveal the irreversible non-affine response of the heap. We measure the spatial distribution of the non-affine strain field for deformations of varying amplitude.

5:37PM E35.00005 Dispersion of Suspensions in Unsteady Microchannel Flows , MARTIN MAXEY, AMANDA HOWARD, LUKAS WINKLERPRINS, ANUBHUV TRIPATHI, Brown University, KYONGMIN YEO, IBM Watson Research Center — We explore the dispersion of non-Brownian (Pe $>>$1) suspensions in unsteady, low Reynolds number shear flows in a microchannel. Prior experimental work on oscillating Couette flows and Poiseuille flows has shown the importance of strain amplitude in determining the long term distribution of particles across the channel. We will present results from numerical simulations for the early development of these flows and the motion of finite length suspension plugs. The distortion of a plug by the shear flow results in inhomogeneous particle fluxes across the channel. This is largely reversible over the course of a full cycle, giving reversibility in the bulk. Self-diffusion gives irreversibility though at the microscale. As the strain amplitude increases or the initial volume fraction increases irreversibility in the bulk is seen. The dynamics behind these processes and the role of particle pressure will be noted, together with related experimental observations.

Sunday, November 24, 2013 4:45PM - 6:03PM –

Session E36 Waves I 407 - Sagnik Mazumdar, University of Pittsburgh

4:45PM E36.00001 Wavemaking by a vortex pair in stratified flow , SURUPA SHAW, JOHN MCHUGH, University of New Hampshire — Recent simulations of a vortex pair in a stratified fluid show that for small Froude number W/Nb the vortices disintegrate into internal waves, where W is the vortex strength, b is the vortex spacing, and N is the buoyancy frequency. The kinetic energy loss from the vortex pair in this regime can be remarkably fast, essentially annihilating the coherent vortex pair before any noticeable propagation. If the Froude number is large the vortices remain coherent and propagate as they would in constant density flow. The transition in behavior occurs near a Froude number of unity, but is apparently not a sharp transition, as some wave-making appears to happen for Froude numbers above unity. Here we quantify the wave-making with an integral of the momentum flux around a sequence of circles centered on the vortex pair and moving with it. Numerical solutions are obtained using a spectral method, the flow is treated as Boussinesq and viscous, and the initial conditions are approximately the flow due to a line vortex. The results confirm that the transition is gradual, although the complexity of the wavy flow makes interpretation difficult. These results are related to vortex roll-up in a stratified fluid.

4:58PM E36.00002 Resonance Van Hove Singularities in Weak Wave Turbulence , YI-KANG SHI, GREGORY EYINK, The Johns Hopkins University — Wave kinetic theory has been developed by Hasselmann, Benney, and others to describe turbulence of weakly nonlinear, dispersive waves. However, systems which are generally dispersive can have resonant sets of wave modes with identical group velocities, leading to a local breakdown of dispersivity. This shows up as a geometric singularity of the resonant manifold and possible divergence of the phase measure in the collision integral. Such singularities occur widely, including acoustical waves, Rossby waves, helical waves in rotating fluids, etc. These singularities are the exact analogue of the critical points found by Van Hove in 1953 for phonon dispersion relations in crystals. The importance of these singularities in wave kinetics depends on the dimension of phase space D=(N-2)d (d physical space dimension, N the number of waves in resonance) and the degree of degeneracy of the critical points. Following Van Hove, we show that point (non-degenerate) singularities produce no divergences for D>2 but lead to logarithmic divergences when D=2 and possible breakdown of wave kinetics. These divergences are not removed by nonlinear broadening of the resonances. We discuss an example of Michel et al. (2010) for optical turbulence (N=4,d=1). When D>2, breakdown can occur for degenerate critical points which live on higher-dimensional lines and surfaces. We give examples for inertial waves in rotating fluids (N=3,d=3) and the dilute electron-hole plasma in graphene (N=4, d=2).

5:11PM E36.00003 Numerical study of nonlinear full wave acoustic propagation[1] , ROBERTO VELASCO-SEGURA, PABLO L. RENDON, Centro de Ciencias Aplicadas y Desarrollo Tecnologico, UNAM — With the aim of describing nonlinear acoustic phenomena, a form of the conservation equations for fluid dynamics is presented, deduced using slightly less restrictive hypothesis than those necessary to obtain the well known Westervelt equation. This formulation accounts for full wave diffraction, nonlinearity, and thermoviscous dissipative effects. A CLAWPACK based, 2D finite-volume method using Roe's linearization has been implemented to obtain numerically the solution of the proposed equations. In order to validate the code, two different tests have been performed: one against a special Taylor shock-like analytic solution, the other against published results on a HIFU system, both with satisfactory results. The code is written for parallel execution on a GPU and improves performance by a factor of over 50 when compared to the standard CLAWPACK Fortran code. This code can be used to describe moderate nonlinear phenomena, at low Mach numbers, in domains as large as 100 wave lengths. Applications range from modest models of diagnostic and therapeutic HIFU, parametric acoustic arrays, to acoustic wave guides. A couple of examples will be presented showing shock formation and oblique interaction.

[1]DGAPA PAPIIT IN110411, PAEP UNAM 2013

5:24PM E36.00004 Wave propagation in a viscous fluid with a pipeline shear mean flow and application for ultrasonic flow meter , YONG CHEN, YIYONG HUANG, XIAOQIAN CHEN, Institute of Space Technology, College of Aerospace Science and Engineering, National University of Defense Technology — This paper deals with the problem of wave propagation in a compressible viscous fluid confined by a rigid-walled circular pipeline in the presence of a shear mean flow. On the assumption of isentropic and axisymmetric wave propagation, the convected acoustic equations are mathematically deduced from the conservations of continuity and momentum, leading to a set of coupled second-order differential equations with respect of the acoustic pressure and velocity components in radial and axial directions. A solution based on the Fourier-Bessel theory, which is complete and orthogonal in Lebesgue space, is introduced to transform the differential equations to an infinite set of homogeneous algebraic equations, thus the wave number can be calculated due to the existence condition of a non-trivial solution. After the discussion of the method's convergence, the cut-off frequency of the wave mode is theoretically analyzed. Furthermore, wave attenuation of the first four wave modes due to fluid viscosity is numerically studied in the presence of the laminar and turbulent flow profiles. Meanwhile, the measurement performance of an ultrasonic flow meter based on the difference of downstream and upstream wave propagations is parametrically addressed.

5:37PM E36.00005 Forced Convective Thermal Transport and Flow Stability Characteristics in Near-Critical Supercritical Fluid , NUSAIR HASAN, BAKHTIER FAROUK, MEM Department, Drexel University, Philadelphia PA 19104 — Forced convective thermal transport characteristics of supercritical carbon dioxide in vertical flow are numerically investigated. A tube with a circular cross-section and heated side-wall is considered. A real-fluid model for representing the thermo-physical properties of the supercritical fluid along with the fully compressible form of the Navier–Stokes equations and an implicit time-marching scheme is used to solve the problem. Thermo-physical properties of near-critical supercritical fluids show diverging characteristics. Large variations of density of near-critical supercritical fluid in forced convective flow can induce thermo-hydraulic instability similar to *density wave oscillations*. The developed numerical model is used for studying the effect of geometrical parameters of the tube, wall heat flux and pressure on steady-state convective thermal transport as well as the stability behavior of the supercritical fluid near its critical point. The enhancement or deterioration of heat transfer caused by the temperature-induced variation of physical properties (especially specific heat) is also investigated, as well as the effect of buoyancy on the forced convective flow.

5:50PM E36.00006 ABSTRACT WITHDRAWN —

6:15PM - 6:15PM —

Session 1A Student Poster Session (6:15PM-7:00PM) South Terrace Foyer, 3rd Floor -

1A.00001 ENERGY AND APPLICATIONS — .

1A.00002 The Study of a Liquid Droplet Falling Through Two Immiscible Layers of Liquids , BIANCA MESA[1], Florida Atlantic University — In an exploratory experiment, we noticed the unusual behaviors of liquid droplets falling through layers of oil and water. A rectangular container was filled with an aqueous solution and a layer of oil. A dropper was used to control the size of the droplet. Water was mixed with Bromothymol Blue dye, a chemical indicator, to visualize the flow processes. Surface tension and the buoyancy of the oil layer between the liquid droplet and the water below caused the liquid droplet to be stopped at the interface. Over time, the support weakened and the droplet would fall quickly through the water. The first of two cases was a salt water solution with NaOH, and the second consisted of balsamic vinegar and NaOH. Once the salt water droplet touched the aqueous solution, it collapsed, sank and spread rapidly at the interface. The sinking motion dragged the spreading fluid back to its center and then down. For the second case, a trace amount of the droplet spread rapidly at the interface while the main portion of the droplet sank and then spontaneously exploded. The difference in behavior is mainly due to the surface tension of the droplet in water. The underlying mechanisms of the droplet's flow instability are from the effects of diffusion weakening the surface tension.

[1]Bianca Mesa is an undergraduate student in the Ocean and Mechanical Engineering Department at Florida Atlantic University. She is pursuing a B.S. degree in Ocean Engineering. In addition to her academic interests, she is also an avid sailor.

1A.00003 The Effect of Magnetic Fields on the Capture of Magnetic Nanoparticles[1] , CHELSEA FUJINAKA, Milwaukee School of Engineering, CHRIS BRAZEL, RHYTHM SHAH, The University of Alabama — It is hypothesized that magnetic nanoparticles may be used in active targeting cancer treatment by localizing the particles in the area of the tumor. To test this hypothesis static and electromagnetic fields were applied to a flow system, and UV-VIS spectroscopy was used to calculate the percentage of particles captured. Uncoated Maghemite nanopowder and FluidMAG-PAD (Chemicell) nanoparticles coated with polyacrylamide were used. Dynamic light scattering was used to look at particles size with and without proteins. Proteins caused the uncoated particles to aggregate. The static field captured approximately 15% of the maghemite nanoparticles in water in a flowing at 0.1 mL/s when using two neodymium magnets laid lengthwise along 2 mm inner diameter tubing. The electromagnetic field pulled the uncoated particles out of the dispersion, but did not capture any in one place. The FluidMAG-PAD particles could not be pulled out of solution by the static field or the electromagnetic field. In order to effectively treat cancer, nanoparticles with a coating would have to be used to avoid opsonization and aggregation within the blood stream; however they cannot be so well dispersed as to not be affected by the magnetic field. The uncoated particles exhibited the capture desired, but do not interact well with proteins. A stronger magnetic field may allow the same capture of the coated particles, but it may also be important to look for a dispersion of nanoparticles not quite as well dispersed as the FluidMAG-PAD.

[1]Support from NSF grant #1062611 is gratefully acknowledged.

1A.00004 Using Thermo-Responsive, Fiber-filled Gels to Control Droplet Motion , GERALD MCFARLIN IV, XIN YONG, OLGA KUKSENOK, ANNA BALAZS, University of Pittsburgh — Using a polymeric gel and elastic fibers, we design a thermo-responsive composite film that can be harnessed to manipulate the droplet motion in microfluidic devices. At low temperatures, the fibers are hidden and unable to interact with external fluid. At higher temperatures, the gel shrinks and exposes the fibers to the external solution; hence the exposed fibers can be utilized to hinder the motion of fluid-driven droplets on the film surface. We use dissipative particle dynamics (DPD) to model our system. We construct the gel in a coarse-grained manner by crosslinking polymer chains. We examine the volume phase transition and swelling kinetics of the gel in explicit solvents and validate our model through comparisons with Flory-Huggins theory. During simulations, a hydrophobic droplet is introduced to the outer solvent and driven over the film surface by an external flow. We focus on the effects of the imposed flow, temperature variations, and droplet-fiber interactions on the droplet's motion. We show that by varying the temperature of the system, we can program the film to interact with the droplet in a well-controlled manner. Our findings reveal how nanofibers can be used to enhance the properties of thermo-responsive gel coatings.

1A.00005 Effective Collecting Method of Volatile Organic Compounds in Water by Bubbling , HITOSHI KIDA, HAYATO HORI, YUZO NONOGUCHI, MASAHARU KAMEDA, RYOICHI SATO, Tokyo University of Agriculture and Technology — A technique is proposed to improve the collection efficiency of a small amount of volatile organic compounds in the gas by impinger, which is generally used as gas collection device for analysis. Eugenol was used as a simulated substance of explosives. The sample gas containing specific amount of eugenol was collected in pure water by the impinger. The concentration of eugenol in water was measured by gas chromatography. The experimental results shows that the collection efficiency of eugenol by the impinger increased as the water level (volume) increased. The bubble motion in the impinger observed by high-speed photography indicates that the averaged values of equivalent diameter and rising velocity of bubbles were reduced as the water level increased. This reduction yields the increase of the resident time of bubble per unit volume of water, which enhances the dissolution of eugenol. On the basis of these characteristics, small glass beads were stuffed into the impimger to increase the resident time per unit volume. The collection efficiency was improved by stuffing the glass beads. Now we test the odorant binding protein as additive for further improvement of collection efficiency.

1A.00006 Electrospray of Solution Processed Nanomaterials , NICHOLAS BROWN, PAUL CHIAROT, State University of New York at Binghamton — Electrospray is a technique that uses large electric fields to generate a spray of highly-charged, monodispersed droplets from a liquid solvent. Colloidal inks, consisting of nanoparticles dispersed in a volatile solvent, can be atomized using electrospray. In this study, we investigate the deposit structure of nanoparticle inks printed onto three different substrates: bare glass, silanized-patterned glass, and glass coated with a liquid film. The deposition morphology of colloidal inks printed onto these surfaces is predicted using mathematical modeling and statistical analysis. The goal of intervening at the substrate with surface patterns and liquid films is to exert control over the microstructure of the printed deposit. The advantage of electrospray is that it is an additive process which drastically reduces material waste that is inherent in other thin-film material processes.

1A.00007 International Senior Design Service Learning Project: Creating a Water System for Kuna Nega in Panama City, Panama , DAN BUDNY, University of Pittsburgh — International service-learning projects are an effective educational tool for universities striving to meet the ABET engineering criterion, while also providing transformational experiences to their students and a service to needy populations in the world. This student poster discusses the benefits of an international service-learning project in Panama City, Panama. The presentation will discuss the design and installation of a water distribution system including a two pressure system, two ground storage tanks, a pump station and the various control systems to fill the tanks. To meet the water demand with the limited supply additional individual rain water collection systems were also installed at individual houses to provide a gray water system for bathing. The year-long process of development design and construction will be described and how it fits within the Swanson School of Engineering Department of Civil Engineering senior design course. This project was a collaboration between the senior design course, and a local chapter of Engineers Without Borders.

1A.00008 Aerodynamics Simulations for the D8 "Double Bubble" Aircraft Using the LAVA Unstructured Solver[1] , SEAN BALLINGER, Columbia University School of Engineering and Applied Science — The D8 "double bubble" is a proposed design for quieter and more efficient domestic passenger aircraft of the Boeing 737 class. It features boundary layer-ingesting engines located under a non-load-bearing π-tail and a lightweight low-sweep wing for flight around Mach 0.7. The D8's wide lifting body is expected to supply 15% of its total lift, while a Boeing 737's fuselage contributes only 8%. The tapering rear of the fuselage is also predicted to experience a negative moment resulting in positive pitch, produce a thicker boundary layer for ingestion by distortion-tolerant engines, and act as a noise shield. To investigate these predictions, unstructured grids generated over a fine surface triangulation using Star-CCM+ are used to model the unpowered D8 with flow conditions mimicking those in the MIT Wright brothers wind tunnel at angles of attack from -2 to 14 degrees. LAVA, the recently developed Launch Ascent and Vehicle Aerodynamics solver, is used to carry out simulations on an unstructured grid. The results are compared to wind tunnel data, and to data from structured grid simulations using the LAVA, Overflow, and Cart3D solvers.

[1]Applied Modeling and Simulation Branch, NASA Advanced Supercomputing Division, funded by New York Space Grant

1A.00009 Improving wind turbine array efficiency through active flow control , JOHN-MICHAEL VELARDE, GUANNAN WANG, PATRICK SHEA, MARK GLAUSER, Syracuse University, LUCIANO CASTILLO, Texas Tech University — We attempted to demonstrate the capability of instrumenting three wind turbine blades with an air delivery system that provided active flow control in an effort to improve turbine performance in the presence of the wake turbulence that is inherent in a turbine array. Presently, turbines are being designed for set conditions, such as steady incoming wind and a set velocity profile, however conditions can be drastically different in the field – thus causing poor performance from the turbines. The blades were instrumented with pressure transducers which measured the suction surface pressure; the sensor setup was such that three unique blade configurations existed: spanwise sensors, chord-wise sensors, and a reference sensor. The compressed air was delivered through a rotary union connected to the turbine hub with tubing attached to the suction side of the blades. The primary purpose of this test was to demonstrate the ability to deliver air to a rotating frame for active flow control. We collected data under three test conditions using an open-section wind tunnel, courtesy of Texas Tech University: static with no flow control, rotation with no flow control, and rotation with active flow control.

1A.00010 Optimal Hydrofoil Kinematics for Tidal Energy Extraction , SARAH FRANK, SHREYAS MANDRE, JENNIFER FRANCK, Brown University — The pitch and heave kinematics of an oscillating hydrofoil are explored for tidal energy extraction using 2D Direct Numerical Simulation in a non-inertial reference frame. The hydrofoil is modeled by an ellipse of aspect ratio 10 at a Reynolds number of 1000 in uniform freestream. Starting with sinusoidal motion in pitch and heave, the heaving magnitude, pitch angle, frequency, and phase angle between pitch and heave were varied. The optimal case had a maximum heave of .5 chord lengths, a maximum pitch angle of 75 degrees, a non-dimensional frequency of 0.15, and a phase of 90 degrees, which are consistent with similar computational studies, and parallel theory and experimentation. In order to further optimize the hydrofoil's stroke for fluid energy extraction, higher harmonics are systemically added to the kinematics, finding that small perturbations to the heave signal can increase the efficiency by up to 6.0%.

1A.00011 Development of a Burner System and Rayleigh Scattering Method to Measure Soot Concentration for Diesel-Relevant Fuels[1] , SARA FLETCHER, Washington University in St Louis, BRIAN FISHER, University of Alabama — Soot, a harmful component of particulate matter, is found in high concentrations in diesel exhaust. This work aims to develop a better understanding of the relationship between chemical structure and soot evolution, which is expected to inform methods to reduce or eliminate soot in diesel combustion. Successful aspects of previous experiments have been combined into a new method to characterize soot formation, growth, and oxidation. Soot is quantified via combined Rayleigh scattering and extinction, using a pulsed 532-nm Nd:YAG laser and sensitive photodetectors. A methane/oxygen diffusion flame serves as a baseline, then species of interest are doped into the fuel stream in low concentration and the change in soot is quantified relative to the base flame. This perturbation method enables study of soot for different species in a flame that has nominally constant global properties. This study focused on fuel components n-heptane and toluene, which have straight-chain and aromatic molecular structures, respectively. Soot was quantified throughout the flame, and it was found that the soot scattering signal was significantly higher for toluene than for n-heptane. Analysis of the signals to quantify actual soot concentrations remains a topic of future work.

[1]Funding from NSF REU grant 1062611

1A.00012 Parameters of a Steady State Model for In-Cylinder Flow of an Internal Combustion Engine[1] , ELIZABETH FORTNER, Worcester Polytechnic Institute, PAUL PUZINAUSKAS, NICHOLAS BOLUS, University of Alabama — Flow structures in an internal combustion engine are critical to engine performance and fuel consumption. Experiments are often conducted to explore how intake port geometry can be modified to induce desired tumble and swirl flow structures within the cylinder. To make these experiments cost-effective, they are often first conducted using a model cylinder on a steady flow bench prior to, or in lieu of, performing full unsteady engine tests. This research examines how model characteristics and experimental configuration choices affect results on these steady-flow tests. The experimental set-up uses DPIV to visualize the flow and a horizontally extracting swirl meter to measure the strength of the tumble structure. The configurations and characteristics examined included model geometry, seeding particle type and location of flow induction. The symmetric geometry experiment investigates how extraction affects the flow structures inside the cylinder. Three different seeding particles were used to see how particle properties affect DPIV results. Reversing the direction of flow through the system causes set-up challenges with removing leaks and introducing seeding particles, but is safer as it directs particles away from the flow bench. Deviation of results from the different test set-ups may indicate that cylinder model experiments need to be carefully designed to ensure high quality results accurate enough for use in designing full scale engine tests.

[1]Support from NSF REU Grant #1062611 is gratefully acknowledged.

1A.00013 JETS, ACOUSTICS AND SHOCKWAVES — .

1A.00014 Characterization of Noise and Instability in a Commercial Burner[1] , STEWART CARPENTER, Case Western Reserve University, AJAY AGRAWAL, The University of Alabama — A range of combustion applications produce noise as a significant and undesirable output. Concurrently, efforts to reduce emissions through lean premixed combustion have shown this process to be prone to developing instabilities. In this study a commercial-style combustor was investigated to characterize combustion noise and instabilities. Knowledge in this area is intended for future research involving the application of porous inert media (PIM) in industrial burners. Porous media has been used to passively suppress both combustion noise and instabilities in a laboratory setting, but has yet to be implemented in a commercial burner. Combustion experiments were conducted in an industrial-scale lean premixed burner using natural gas while varying equivalence ratio and reactant flow rate. Acoustic data was acquired using a microphone probe placed in the plane of the combustor exit. Measurements were analyzed in the frequency spectrum to quantify noise spectra and detect the development of instabilities. Results have indicated the occurrence of strong combustion instability at certain conditions. Additionally, research has supported the general relationship of increased noise production with increasing equivalence ratio and heat release rate. Adverse effects of combustion instability were accompanied with flashback and downstream acoustic excitation.

[1]Funding for this research provided by NSF REU grant 1062611.

1A.00015 Identifying Potential Noise Sources within Acoustic Signals , VICTORIA HOLCOMB, JACQUES LEWALLE, Syracuse University — We test a new algorithm for its ability to detect sources of noise within random background. The goal of these tests is to better understand how to identify sources within acoustic signals while simultaneously determining the strengths and weaknesses of the algorithm in question. Unlike previously published algorithms, the antenna method does not pinpoint events by looking for the most energetic portions of a signal. The algorithm searches for the ideal lag combinations between three signals by taking excerpts of possible events. The excerpt with the lowest calculated minimum distance between possible events is how the algorithm identifies sources. At the minimum distance, the events are close in time and frequency. This method can be compared to the cross correlation and denoising methods to better understand its effectiveness. This work is supported in part by Spectral Energies LLC, under an SBIR grant from AFRL, as well as the Syracuse University MAE department.

1A.00016 Comparison of Methods for Identifying Noise Sources in Far-Field Acoustic Signals , ANDREW TENNEY, JACQUES LEWALLE, Syracuse University — Three different methods of extracting intermittent wave packets from unstructured background within complex time series signals were analyzed and compared. The algorithms are denoted "cross correlation," "denoising," and "TFLE (Time-Frequency-Lag event)" methods respectively. All three methods utilize Mexican Hat or Morlet wavelets for the transformation of time domain signals into time-frequency domain signals. Within the denoising and cross correlation algorithms, events are identified through comparison of high energy excerpts of each signal captured by individual far-field microphones, while the TFLE algorithm simply defines events by their contributions to positive correlation values. The goal of this analysis is to quantify the advantages and disadvantages of each of these methods. The results lend themselves to determining the validity of these methods as noise source identification algorithms to be used in jet noise characterization. This work is supported in part by Spectral Energies LLC, under an SBIR grant from AFRL; and by the Department of Mechanical and Aerospace Engineering REU Program at SU.

1A.00017 ABSTRACT WITHDRAWN —

1A.00018 Shock wave reflections in a liquid filled thin tube , SHOTA YAMAMOTO, YOSHIYUKI TAGAWA, MASAHARU KAMEDA, Tokyo University of Agriculture and Technology — We investigate a behavior of an underwater shock wave in a thin glass tube using an ultra high-speed camera up to 10^7 frames per second. We here focus on the pressure of the reflected shock wave at interfaces (water-glass wall / water-air). A shock wave is visualized using the Background Oriented Schlieren (BOS) technique. We measure the time evolution of the shock front position and estimate the shock velocity, pressure, and internal energy as a function of the distance from the shock center. At the water-wall interface the reflected shock pressure is lower than the incident shock pressure, which agrees well with the theoretical estimation for an acoustic pressure wave. The reflected pressure at the air-water interface is much lower than the incident shock, indicating that the shape of the air-water interface may affect this reduction of the reflected pressure.

1A.00019 A focused liquid jet using a pressure impulse , AKIHITO KIYAMA, YUTO NOGUCHI, YOSHIYUKI TAGAWA, Tokyo University of Agriculture and Technology — We examine a behavior of a focused liquid jet using a pressure impulse inside a test tube filled with a wetting liquid. It is found that the jet velocity depends on the initial height of the test tube while it is independent of the diameter of the tube. We rationalize these results by employing a pressure impulse approach and considering a flow focusing due to the concave shape of the meniscus. In addition, we generate the focused liquid jet consisting of two different liquids. The jet velocity varies non-monotonically with the ratio of the two liquids.

1A.00020 A new classification for liquid jets dynamics , BOWEN LING, ILENIA BATTIATO, Clemson University — The physics of liquid jets has been attracting scientists' interest for many decades. Previous works have focused on various aspects of jets dynamics including instability, self-similarity, etc. We propose a new criterion to classify liquid jets dynamics based on a non-dimensionalization of Navier-Stokes equations, which generalises well-established scalings. We employ such framework to describe specific dynamics, e.g. breakup characteristics, drop formation and dripping-jetting transition, and identify the driving physical mechanisms of different regimes. We compare the proposed classification with experimental results.

1A.00021 Characterization of synthetic jet actuators used for jet noise reduction by flow control , ALEXIS ZELENYAK, ZACHARY BERGER, MATTHEW BERRY, PATRICK SHEA, MARK GLAUSER, Syracuse University — The issue of jet noise introduces various opportunities for advancements in flow control and fluid dynamics. One such method for jet noise reduction involves the use of synthetic jet actuators as shear layer excitation on the flow produced by a fully compressible, turbulent jet. A set of eight zero-net-mass flux actuators are organized around the periphery of the jet in an actuation glove fitting on the nozzle. As some noise reduction has been achieved through the use of this actuation system, further characterization of the system is necessary to fully quantify its capabilities and understand its effect on the flow physics in the shear layer. The synthetic jet actuators are driven by several different frequencies based on the Helmholtz resonance of the cavities, with measurements taken at several locations along the actuator orifice. Velocity profiles are then constructed from the measured response using hot wire anemometry. Such experimental results provide vital insight into the flow field created by the synthetic jet actuator system, allowing for more effective modification to the actuation glove.

1A.00022 Heater Applications for High Speed Jets , JACK ROSSETTI, Buffalo, ZACHARY BERGER, MATTHEW BERRY, Syracuse, ANDRE HALL, Pratt & Whitney, MARK GLAUSER, Syracuse — In this investigation, we study a high speed jet flow for noise reduction techniques. Here we specifically examine a heated jet for practical jet noise applications. Experiments are conducted in the Syracuse University anechoic chamber at the Skytop campus. This 206 m3 facility is lined with fiberglass wedges having a cutoff frequency of 150 Hz. Far-field microphones and near-field pressure sensors measure the acoustics and hydrodynamics, respectively. A 470 kW Chromalox heating unit is used to heat the flow to 1000°F at the nozzle exit. The controller for the heating unit has an associated time lag based on the Mach number and temperature. Therefore, this study will primarily focus on the heat transfer between the heating elements and the nozzle flow. Optimization of the heater's controller will allow for sufficient run time for data acquisition capabilities. Previous investigations at Syracuse University indicate significant differences between heated and cold jets, with regards to the acoustics and potential core characteristics (Hall et al. 2009).

1A.00023 Statistical Comparison of Far-Field Noise Events in a Controlled Flow Ma=0.6 Jet , GRAHAM FREEDLAND, JACQUES LEWALLE, Syracuse University — We compare distributions of acoustic events in controlled and uncontrolled high speed jets. By examining far-field acoustic signals from three microphones and using continuous wavelets, sources of noise can be identified through cross-correlation of the different far-field signals. From the events found, four properties (wavelet magnitude, Strouhal number and lags between two pairs of microphones) were tabulated. Each test gives over 10,000 events, which were sorted into histograms that approximate the statistical distributions of properties. This is used to determine what influence the addition of synthetic jet actuators has on the properties of the flow of the jet. A qualitative analysis of the distributions using quantile-quantile plots helps in the understanding of the distributions of sources. A quantitative analysis using the Anderson-Darling and Kolmogorov-Smirnov tests establishes statistically significant differences between the baseline and control cases. The authors thank Dr. Mark Glauser, Dr. Kerwin Low and the Syracuse Jet Group for the use of their data, Professor Dongliang Wang of Upstate Medical University for his suggestion of statistical methods, and Spectral Energies LLC (through an SBIR grant from AFRL) for their support.

1A.00024 Focusing of cylindrical liquid jets into droplets , KRISTEN EDWARDS, AMY MCCLENEY, PHILIPPE BARDET, The George Washington University — Upward angled water jets discharging in quiescent air are studied experimentally with time varying forcing. The jets issue from a 2 mm diameter tube, while highly controllable forcing is accomplished with a magnetic linear motor coupled with an arbitrary waveform generator. In particular, regimes of jet focusing are generated at various injection rates. The jets result in large droplets that can be created at various elevations. This type of flow mimics the spray generated by an Archer fish. Actual forcing functions were monitored using LDT.

1A.00025 BIOLOGICAL, MICRO AND COMPLEX FLUIDS — .

1A.00026 Thin-Film Drainage and Droplet Adhesion in a Microfluidic Channel , JONATHAN HUI, WEI WANG, PETER HUANG, Binghamton University — In many multiphase fluid processes, such as in petroleum extraction and biochemical analysis, one often sees the lodging of immiscible droplets that block flow in a conduit. The absence of a thin-film lubrication layer surrounding adhered droplets significantly increases the threshold pressure gradient required to induce bulk flows. In this work, we investigate the thin-film drainage process that leads to droplet adhesion and study how coating droplets with charged surfactants or solid particles can prevent direct contact between the droplets and channel wall. We report on our current computational and experimental results of an oversized gas droplet in a water-filled flow channel under the influence of surface tension and interfacial electrostatic repulsion.

1A.00027 On the heat transfer and flow of a non-homogenous fluid , JOSEPH FIORDILINO, ASHWIN VAIDYA, Department of Mathematical Sciences, Montclair State University, MEHRDAD MASSOUDI, U.S. Department of Energy, National Energy Technology Laboratory — In this study, we consider the flow of a complex fluid such as coal-water slurry or biomass. We assume the suspension can be modeled as a non-homogenous viscous fluid whose viscosity is a function of space and temperature. We study the heat transfer and the steady fully developed flow of this complex fluid between two long horizontal plates subject to the no-slip condition at the plates. Two different correlations are proposed for the viscosity and the thermal conductivity and analytical and numerical results are presented for the velocity, temperature and the volumetric flow rate.

1A.00028 Remote recoil between waves and vortices in superfluids , YUAN GUO, OLIVER BUHLER, Courant Institute of Mathematical Sciences, New York University — This is a theoretical and numerical study of a particular wave-vortex interaction effect in superfluids, which extends previous work in classical compressible fluid mechanics. The fundamental modeling assumption is that both waves and vortices can be described by the defocusing nonlinear Schrödinger equation. At play is the refraction of small-scale waves by inhomogeneous straining flows due to one or several line vortices and the concomitant back-reaction, the"remote recoil", that is felt at the vortex locations. The remoteness is meant to highlight that the waves and the vortices are far from each other, and do not overlap in physical space. This recoil is of second order in wave amplitude and can be computed from the pseudomomentum budget of the waves. The recoil force and the scattering angle are computed both for finite and infinite wavetrains and the results are cross-checked against numerical integration of the relevant ray-tracing equations. We also consider the peculiar case of a wavetrain collapsing onto a single vortex, in which the wave-vortex interactions are not remote anymore. For some parameter values the WKB theory underlying ray tracing may retain its validity during the wave collapse. This would be a novel form of singular absorption of waves by a vortex.

1A.00029 Computational models for fluid-structure interaction with a poroelastic structure , RANA ZAKERZADEH, PAOLO ZUNINO, Department of Mechanical Engineering and Materials Science,University of Pittsburgh, MARTINA BUKAC, IVAN YOTOV, Department of Mathematics ,University of Pittsburgh — In the context of hemodynamics, we model blood flow in arteries as an incompressible Newtonian fluid confined by a multilayered poroelastic wall. We consider a two layer model for the arterial wall, where the inner layers behave as a thin structure modeled as a linearly elastic membrane, while the outer part of the artery is described by the Biot model. We propose and analyze a splitting strategy, which allows solving the Navier-Stokes and Biot equations separately. In this way, we uncouple the original problem into two parts defined on separate subregions, leading to a more efficient calculation of the numerical solution. The theoretical results will be complemented by numerical simulations. We numerically investigate the effects of porosity to the structure displacement. Namely, we distinguish a high storativity and a high permeability case in the Darcy equations, and compare them to the results obtained using a purely elastic model. A physical interpretation of the observed phenomena will be discussed. Indeed, the role of the proroelastic parameters on the pressure wave propagation in arteries emerges from the analysis of an equivalent formulation of the Biot system, where all the equations are condensed into a single one.

1A.00030 Engineered Asymmetric Synthetic Vesicles , LI LU, PAUL CHIAROT, State University of New York at Binghamton — Synthetic vesicles are small, fluid-filled spheres that are enclosed by a bilayer of lipid molecules. They can be used as models for investigating membrane biology and as delivery vehicles for pharmaceuticals. In practice, it is difficult to simultaneously control membrane asymmetry, unilamellarity, vesicle size, vesicle-to-vesicle uniformity, and luminal content. Membrane asymmetry, where each leaflet of the bilayer is composed of different lipids, is of particular importance as it is a feature of most natural membranes. In this study, we leverage microfluidic technology to build asymmetric vesicles at high-throughput. We use the precise flow control offered by microfluidic devices to make highly uniform emulsions, with controlled internal content, that serve as templates to build the synthetic vesicles. Flow focusing, dielectrophoretic steering, and interfacial lipid self-assembly are critical procedures performed on-chip to produce the vesicles. Fluorescent and confocal microscopy are used to evaluate the vesicle characteristics.

1A.00031 Fluid Mechanics of the Vascular Basement Membrane in the Brain , MIKHAIL COLOMA, JONATHAN HUI, PAUL CHIAROT, PETER HUANG, State University of New York at Binghamton, ROXANA CARARE, University of Southampton, KENNETH MCLEOD, DAVID SCHAFFER, State University of New York at Binghamton — Beta-amyloid is a normal product of brain metabolic function and is found within the interstitial fluid of the brain. Failure of the clearance of beta-amyloid from the aging brain leads to its accumulation within the walls of arteries and to Alzheimer's disease. The vascular basement membrane (VBM) within the walls of cerebral arteries surrounds the spirally arranged smooth muscle cells and represents an essential pathway for removal of beta-amyloid from the brain. This process fails with the stiffening of arterial walls associated with aging. In this study we hypothesize that the deformation of the VBM associated with arterial pulsations drives the interstitial fluid to drain in the direction opposite of the arterial blood flow. This hypothesis is theoretically investigated by modeling the VBM as a thin, coaxial, fluid-filled porous medium surrounding a periodically deforming cylindrical tube. Flow and boundary conditions required to achieve such a backward clearance are derived through a control volume analysis of mass, momentum, and energy.

1A.00032 A simple microfluidic-inspired extensional flow device for observation of small aquatic organisms: design and implementation , NEIL THOMAS, RACHEL PEPPER, DORIAN LIEPMANN, M.A.R. KOEHL, UC Berkeley — We present a simple method for trapping microscopic particles and organisms around 100 μm in size. Using a laser-cut acrylic device and gravity-fed flow, single particles or organisms are trapped in a stagnation point formed at the center of intersecting channels, where they are observed under a microscope. Objects can be trapped by controlling the flow along the extensional axis, which is achieved by varying the outflow rate of one exit channel. We show results from applying this method to study the response of marine larvae (of the sea slug, *Phestilla sibogae*) to varying accelerations. We also present details of the simple and inexpensive fabrication technique used to create such small devices. Overall, this fabrication technique allows for the generalization of microfluidic devices to micro- and millimeter scale applications.

1A.00033 Mathematical Model and Simulation of Particle Flow around Choanoflagellates Using the Method of Regularized Stokeslets[1] , NITI NARARIDH, Trinity University — Choanoflagellates are unicellular organisms whose intriguing morphology includes a set of collars/microvilli emanating from the cell body, surrounding the beating flagellum. We investigated the role of the microvilli in the feeding and swimming behavior of the organism using a three-dimensional model based on the method of regularized Stokeslets. This model allows us to examine the velocity generated around the feeding organism tethered in place, as well as to predict the paths of surrounding free flowing particles. In particular, we can depict the effective capture of nutritional particles and bacteria in the fluid, showing the hydrodynamic cooperation between the cell, flagellum, and microvilli of the organism.

[1]Funding Source: Murchison Undergraduate Research Fellowship

1A.00034 Surface Patterning: Controlling Fluid Flow Through Dolphin and Shark Skin Biomimicry[1] , LAWREN GAMBLE, Smith College, AMY LANG, MICHAEL BRADSHAW, ERIC MCVAY, University of Alabama — Dolphin skin is characterized by circumferential ridges, perpendicular to fluid flow, present from the crest of the head until the tail fluke. When observing a cross section of skin, the ridges have a sinusoidal pattern. Sinusoidal grooves have been proven to induce vortices in the cavities that can help control flow separation which can reduce pressure drag. Shark skin, however, is patterned with flexible scales that bristle up to 50 degrees with reversed flow. Both dolphin ridges and shark scales are thought to help control fluid flow and increase swimming efficiency by delaying the separation of the boundary layer. This study investigates how flow characteristics can be altered with bio-inspired surface patterning. A NACA 4412 hydrofoil was entirely patterned with transverse sinusoidal grooves, inspired by dolphin skin but scaled so the cavities on the model have the same Reynolds number as the cavities on a swimming shark. Static tests were conducted at a Reynolds number of approximately 100,000 and at varying angles of attack. The results were compared to the smooth hydrofoil case. The flow data was quantified using Digital Particle Image Velocimetry (DPIV). The results of this study demonstrated that the patterned hydrofoil experienced greater separation than the smooth hydrofoil. It is hypothesize that this could be remediated if the pattern was placed only after the maximum thickness of the hydrofoil.

[1]Funding through NSF REU grant 1062611 is gratefully acknowledged

1A.00035 Vortex Formation on a Plunging Plate with Butterfly Inspired Surface Patterning[1] , PRESTON POWELL, AMY LANG, MICHAEL BRADSHAW, University of Alabama — Previous research has shown that butterfly wings are covered in scales (100 microns in length) that are aligned in rows. When these scales are removed, butterflies require more energy and flaps per second to fly. These scales are pivotal to a butterfly's flying efficiency and are the inspiration for this study. This study examined whether improved efficiency is a result of the prolonged attachment of the leading-edge vortex (LEV) due to the arrangement of these rows. Efficiency is important for any system, however, this type of flight is quite similar to that of an MAV. A long flat plate was lifted at various Reynolds numbers to generate tip vortices on the bottom side. Three test plates were used: one flat plate as a control, one with length-wise ridges, and one with width-wise ridges. These ridges act as a simplistic model of butterfly scales while maintaining flow similarity. DPIV was used to measure the circulation and attachment of the leading-edge vortex for each plate. This experiment tested the hypothesis that the width-wise ridges will exhibit the longest attachment of the LEV which corresponds to increased lift. Also, the plate with length-wise ridges will have the quickest shedding of the LEV and decreased lift.

[1]Funding from NSF REU Grant #1062611 is gratefully acknowledged.

1A.00036 TURBULENCE, STABILITY AND FLUID-STRUCTURE INTERACTION — .

1A.00037 Equilibrium Configurations of a Fiber in a Flow , PAMELA GUERRON, CHRISTOPHER BERGHOUT, BOGDAN NITA, ASHWIN VAIDYA, Department of Mathematical Science, Montclair State University — The aim of this study is to understand the coupled dynamics of flexible fibers in a fluid flow. In particular, we examine the equilibrium configurations of the fiber with changing Reynolds numbers, orientations and lengths of the fiber. Our study is motivated by biological phenomena such as ciliary bending, flexing of plants and trees in winds etc. Our approach to resolving this problem has been threefold: experimental, numerical and theoretical. In our experiments we create physical models of variable length fibers inserted into a basal body structure, which is then suspended in a flow tank and positioned at different angles. The structure (fibers) are subjected to different velocities of water flow, ranging from 0m/s to 0.53 m/s in increments of 0.038 m/s. The results of the experiment were analyzed using Adobe Photoshop and the effect of the above mentioned parameters upon the shape of the fiber is analyzed. In addition, we also simulate this problem using the software Comsol and also create a simple, toy mathematical model incorporating the competing effects of tension and fluid drag on the fiber to obtain a closed form expression. Our various approaches point to consistent results.

1A.00038 Properties of the plasma sheath edge above a rectangular depression in DONUT , THOMAS E. STEINBERGER, T.E. SHERIDAN, Ohio Northern University — Plasma is a gas consisting of positively and negatively charged particles, such as electrons and positive ions. The electric field inside a plasma is very small since plasma is a conductor. When plasma is in contact with a material boundary (i.e., a "wall") a boundary layer called the plasma sheath forms. The electric field inside the sheath is large and points away from the wall. The sheath electric field reduces the loss of highly mobile electrons, while accelerating ions out of the plasma, so that in steady state the electron and ion loss rates balance. The shape of the sheath edge is determined by the shape of the wall and the width of the sheath. We report experimental measurements of sheath conformation to a rectangular depression in a flat horizontal electrode (i.e., "the wall") in the Dusty Ohio Northern University experimenT (DONUT) for various aspect ratios. Clusters of two microscopic dust particles float above the depression at the sheath edge. The horizontal shape of the sheath edge is determined from the horizontal center-of-mass frequencies for the dust particles. The vertical electric field is found from the force balance on the dust particles, and the local charge density is measured using the vertical center-of-mass frequency.

1A.00039 Partially-Averaged Navier-Stokes Modeling of Turbulent Swirling Flow[1] , HOSEIN FOROUTAN, SAVAS YAVUZKURT, The Pennsylvania State University — A Partially-Averaged Navier-Stokes (PANS) model is developed in this study and tested for the simulation of turbulent swirling flow. In the PANS approach, the extent of partial averaging is controlled by the unresolved-to-total ratios of turbulent kinetic energy, k, and turbulent dissipation rate, ε, which depend upon the computational grid spacing. Therefore, PANS closure model can be used at any grid resolution ranging from Reynolds-Averaged Navier-Stokes (RANS) to Direct Numerical Simulation (DNS). The present PANS model is derived from the extended $k-\varepsilon$ turbulence model of Chen and Kim (1987), where an extra time scale of the production is included in the turbulent dissipation rate transport equation. The new model is applied to the simulation of turbulent confined swirling flow through an abrupt expansion with Re=30,000 and swirl number of 0.6. The results are compared to the available experimental data, as well as those obtained using RANS and Detached Eddy Simulation (DES) on the same grid resolution.

[1]Supported by the DOE/PSU graduate student fellowship program for hydropower and the hydro research foundation fellowship.

1A.00040 Pressure Driven Turbulent Flow in a Channel with Superhyrophobic Riblets , ARIAN YOUSEFI, ILENIA BATTIATO, Clemson University — In the past decades, many studies have focused on the ability of micro-patterned surfaces to reduce the fluid resistance in micro-fluidic applications. They revealed that micro-structures treated with hydrophobic coatings can significantly reduce skin drag in both laminar and turbulent regimes. We study pressure driven Navier-Stokes flow through and over a periodic rectangular array of alternating ribs and grooves parallel to the flow direction. The fluids inside and above the grooved surface are air and water, respectively. We employ the method of eigenfunction expansion combined with a domain decomposition approach to obtain a semi-analytical solution for the flow velocity within and above the grooves. The local and mean velocity profiles inside the grooves, the slip length and the slip velocity are determined for a number of different scenarios. Finally, we compare our semi-analytical solution with experimental data.

1A.00041 Hydrodynamic Simulations of Steady-State Density Inversion in Vertically Shaken Granular Layers[1] , FARHEEN SYEDA, JOSH PANFIL, JON BOUGIE, Loyola University Chicago — We investigate density inversion in shaken granular layers using three-dimensional, time-dependent continuum simulations to Navier-Stokes order for a layer of uniform, inelastic, frictionless spheres on a vertically oscillating plate. For given shaking strength, these simulations show cyclic time dependence of the granular layer correlated with the time-dependent oscillation of the plate for low accelerational amplitude. In such cases, the highest density region can be found near the plate during portions of the cycle. When the accelerational amplitude exceeds a critical value, the layer exhibits a steady-state density inversion, in which a high-density region is found far from the plate, supported by a lower-density, gas-like region below. For a variety of dimensionless shaking strengths S, we study the transition from a time-dependent, non-density-inverted state to a steady-state density inversion as a function of the dimensionless accelerational amplitude Γ. In each case, the density profile of the layer exhibits a cyclic oscillation at the driving frequency for low Γ and the response frequency matches the driving frequency through the transition. However, the amplitude of time-dependent response drops as Γ exceeds a critical value.

[1]This research is supported by the Loyola Undergraduate Research Opportunities Program.

1A.00042 Proposed Mechanism for Shock-Driven Stripe Patterns in Vertically Oscillated Granular Systems[1]

, ALEX GILMAN, STEFANIE MOERTL, JON BOUGIE, Loyola University Chicago — We investigate vertically shaken granular systems using numerical solutions of continuum equations to Navier-Stokes order for uniform, frictionless, inelastic spheres. When layers of particles are oscillated at accelerational amplitudes greater than that of gravity, the layers leave the plate, and shocks are created upon re-established contact with the plate. Additionally, standing-wave patterns form when the accelerational amplitude exceeds a critical value. We demonstrate relationships between properties associated with shocks and properties associated with standing wave patterns, and propose a mechanism by which a non-uniform shock front drives standing-wave configurations. For a given layer depth and accelerational amplitude, varying driving frequency alters the shock strength as well as pattern wavelength; increasing layer depth produces stronger shocks and longer wavelengths for a given frequency. We use non-dimensional versions of the Navier-Stokes equations to mathematically derive relationships between these variables. We compare these mathematical relationships to those found empirically through simulations conducted at various layer depths and frequencies.

[1]This research is supported by the Loyola Undergraduate Research Opportunities Program.

1A.00043 ABSTRACT WITHDRAWN —

1A.00044 An Alternative Nondimensional Vibration Frequency for Spanwise Tensioned Membranes in Low Re Flow

, ZHENG ZHANG, ANDREW WRIST, JAMES P. HUBNER, University of Alabama, LAWRENCE UKEILEY, University of Florida — For the fixed flexible membrane wing at low Reynolds number (Re $<$ 100,000), the membrane vibration interacts with the separated shear layer emanating from the leading edge, increasing the momentum transfer and reducing the flow separation. This investigation proposes an alternative vibration frequency scaling for the membrane wing. Compared to the traditional Strouhal scaling, the proposed nondimensional vibration frequency combines effects of the aerodynamically-induced tension, applied tension and membrane properties. A simplified aerodynamically-induced strain model is introduced through assuming uniform aerodynamic loading on the membrane. To verify the vibration frequency scaling and the accuracy of the aero-strain model, high-speed deformation measurement and force measurement of two-dimensional free leading- and trailing- edge membrane wings are performed in the low speed wind tunnel at Re $\sim$ 50,000. The preliminary data show that the proposed scaling is more appropriate than Strouhal scaling when the flow was driven by the membrane motion but not the shedding vortex.

1A.00045 Numerical simulation of Rayleigh-Bernard convection in a cylindrical container[1]

, NORMA Y. SANCHEZ-TORRES, ERICK J. LOPEZ-SANCHEZ, SERGIO HERNANDEZ-ZAPATA, GERARDO RUIZ-CHAVARRIA, Universidad Nacional Autonoma de Mexico — The heat transport by natural convection is a central mechanism in the explanation of many natural phenomena. Despite many works that treat the Rayleigh-Benard convection, most of them describe the phenomenon by making a two-dimensional approach. The purpose of this work is to use a cylindrical geometry. The study further extends to convection driven by evaporation which actually is an open subject. In this work we use a numerical methods to solve the Navier-Stokes, continuity and energy equations: a finite differences method for time, r and z coordinates; and a Fourier spectral method for the angular coordinate. In this manner the numerical code can be parallelized. The boundary conditions are the usual on solid walls, i.e. non-slip for velocity. The system starts at rest. The results are compared with experimental results and data reported in the literature.

[1]Authors acknowledge support by DGAPA-UNAM under project IN116312 “Vorticidad y ondas no lineales en fluidos.”

1A.00046 Simulating the motion of micro-capsules in complex geometries[1]

, LAILAI ZHU, LUCA BRANDT, KTH Mechanics, Linne flow centre — We develop a code to resolve the fluid-structure interaction of capsules in low-Reynolds-number flow, in 3D general geometries. We use an accelerated boundary-integral method, the general geometry Ewald method (GGEM) to solve the Stokes flow in the framework of the Navier-Stokes solver NEK5000 based on the spectral element method. A global spectral interpolation employing spherical harmonics is incorporated simultaneously to resolve the membrane dynamics. Two cases are investigated to illustrate the generality of our implementation. Firstly we simulate a capsule transported in a $3D$ channel and/or duct with a corner, for a better understanding of moving soft objects in geometrically complex configurations. We examine the effect of capsule elasticity and wall confinement in detail. Our results give useful hints for the design of micro-devices. As a second case, we simulate the capsule flowing past a cylindrical obstacle with and without confinement, representing two popular cell separation configurations, pinched flow fractionation (PFF) and deterministic lateral displacement (DLL) respectively. In contrast to the original methodology using fluid inertia, particle size or steric effect, we numerically demonstrate the pure-elasticity-driven cell separation in such devices.

[1]We acknowledge the funding from VR (Swedish research council) and computer time from SNIC

1A.00047 Examining Dynamic Stall for an Oscillating NACA 4412 Hydrofoil[1]

, ERIC MCVAY, AMY LANG, University of Alabama, LAWREN GAMBLE, Smith College, MICHAEL BRADSHAW, University of Alabama — Dynamic stall is unsteady separation that occurs when a hydrofoil pitches through the static stall angle while simultaneously experiencing a rapid change in angle of attack. The NACA 4412 hydrofoil was selected for this research because it has strong trailing edge turbulent boundary layer separation characteristics. General dynamic stall angle of attack for approximately symmetric airfoils has been recorded to occur at 24 degrees, with separation beginning at about 16 degrees. It is predicted that the boundary layer will stay attached at a higher angle of attack because of the cambered geometry of the hydrofoil. It is also hypothesized that the boundary layer separation occurs closer to the trailing edge and that the dynamic stall angle of attack occurs somewhere between 24 and 28 degrees for the oscillating NACA 4412 hydrofoil. This research was conducted in a water tunnel facility using Time Resolved Digital Particle Image Velocimetry (TR-DPIV). The hydrofoil was pitched up from 0 to 30 degrees at Reynolds numbers of 60,000, 80,000 and 100,000. Flow characteristics, dynamic stall angles of attack, and points of boundary layer separation were compared at each velocity with both tripped and un-tripped surfaces. Follow-on research will be conducted using flow control techniques from sharks and dolphins to examine the potential benefits of these natural designs for separation control.

[1]Support for this research by NSF REU Grant #1062611 and CBET Grant #0932352 is gratefully acknowledged.

1A.00048 Interacting Hairpin Vortices[1]

, RIJAN MAHARJAN, DANIEL SABATINO, Lafayette College — The generation mechanism for hairpin vortices is most commonly described as an autogeneration process in which an isolated vortex induces the formation of additional vortices. However, the interaction of two or more vortices can be another generation mechanism. The present work examines the interaction of two artificially generated hairpin vortices in a free-surface water channel. The hairpins are formed by direct injection at different streamwise locations. The different modes of interaction are categorized by the strength and relative position of the vortices at the time of interaction. One of the interaction modes causes the generation of a third hairpin vortex in a process similar to the autogeneration process of isolated hairpins. A comparison of the isolated and interacting generation processes is presented using visualizations and PIV measurements.

[1]Supported by the National Science Foundation under Grant CBET-1040236

1A.00049 VISCOUS FLOWS —

1A.00050 Measurement of traction forces exerted by the foot in motion[1] , MARIA FERNANDA LUGO-BOLANOS, SHREYAS MANDRE, Brown University, MADHUSUDHAN VENKADESAN, National Centre for Biological Science-Tata Institute of Fundamental Research, MAHESH BANDI, Okinawa Institute of Science and Technology Graduate University — When walking and running, the foot acts like a flexible viscoelastic object that cushions impact and stores elastic energy. To characterize the functioning of the foot as a spatially extended and flexible object, we require all components of the ground traction forces to be measured with sufficient spatial and temporal resolution. We present here the theoretical underpinnings of a method based on photoelasticity to measure these traction forces with millimeter scale spatial, and millisecond temporal resolution.

[1]Human Frontier Science Program

1A.00051 The Influence of Dome Size, Parent Vessel Angle, and Coil Packing Density on Coil Embolization Treatment in Cerebral Aneurysms[1] , DAVID H. FRAKES, Department of Biological and Health Systems Engineering; School of Electrical, Computer, and Energy Engineering (Arizona State University, Tempe, AZ), APRINDA INDAHLASTARI, JUSTIN RYAN, M. HAITHEM BABIKER, PRIYA NAIR, VARSHA PARTHAS, Department of Biological and Health Systems Engineering (Arizona State University, Tempe, AZ) — Intracranial aneurysms (ICAs) are dilated cerebral blood vessels. Treating ICAs effectively prior rupture is crucial since their association with 45% mortality rate. Embolic coiling is the most effective ICA treatment. Series of embolic coils are deployed into the aneurysm with the intent of reaching a sufficient packing density (PD) to help seal off the ICA from circulation. While coiling is effective, treatment failures have been associated with basilar tip aneurysms (BTAs), perhaps because of their geometry. The aim of this study was to examine the effect of dome size, parent vessel (PV) angle, and PD on intraaneurysmal (IA) velocity, crossneck (CN) flow and low wall shear stress (WSS) area using simulations and experiments in idealized BTA models. IA velocity and CN flow decreased after coiling, while low WSS area increased. With increasing PD, IA velocity and CN flow were further reduced, but low WSS area had a minimal change. Coil PD had the greatest impact on post-treatment flow while dome size had a greater impact than PV angle. Overall, the role of aneurysmal geometries may vary depending on treatment goal and timing e.g., high coil PD may reduce IA velocity more effectively during early aneurysmal growth when the dome size is small.

[1]Funded by the American Heart Association

6:15PM - 6:15PM –
Session 1B Poster Session (6:15PM- 7:00PM) Spirit of Pittsburgh Gallery -

1B.00001 ACOUSTICS –

1B.00002 Design of an Acoustic Array for Comparison with an Alternative Source Localization Method[1] , DESHAWN COOMBS, JACQUES LEWALLE, MARK GLAUSER, GUANNAN WANG, Syracuse University — We report on the design, testing and construction of a conventional acoustic array, and document an alternate method of signal processing. The purpose of the new algorithm is to improve the spatial localization of acoustic sources. The reference results are obtained using the beamforming algorithm. The array design includes 60 microphones with a maximum aperture diameter of 39 inches. The arrays target frequency range is 500-5000 Hz. The new algorithm uses fewer microphones. We will show results with simulated signals and with jet noise experimental data. Details of the array calibration and representative data from measurements will be presented along with data post-processing procedures.

[1]Support from Syracuse University MAE department and LSAMP.

1B.00003 AERODYNAMICS –

1B.00004 Vortex Generation by a Low-Camber Rotating Arc Wing , MAJID MOLKI, Southern Illinois University Edwardsville — A rotating circular-arc wing is placed in a uniform turbulent flow to generate and study vortices. The momentum equation is modified for the non-inertial rotating reference frame attached to the wing. Turbulence is modeled by the $k-\omega$ SST model. Using the open-source software OpenFOAM, the conservation equations are solved on a dynamic mesh which rotates with the wing, and the flow is resolved all the way to the wall. The computations are performed for Re = 60,000 with rotation number ranging from Ro = 0 to 0.2. Lift and drag coefficients, contours of vorticity and streamlines, and pressure and vorticity over the wing are presented. The relationship between wall vorticity, pressure gradient, and vorticity flux is considered. This study indicates that rotation of the wing creates a dynamic situation that delays the stall to higher angles of attack and enhances the lift and drag coefficients. Depending on the orientation of the wing and rotational speed, a variety of flow patterns appear which include the leading-edge and rolling vortices, dynamic stall, vortex sheets, and stretching and bending of vortex sheets. The relationship between vorticity and pressure gradients are utilized to interpret and explain the flow features.

1B.00005 BIOFLUIDS – .

1B.00006 Low-Reynolds-number swimming near a wall[1] , GAOJIN LI, AREZOO ARDEKANI, University of Notre Dame — Hydrodynamics of swimming organisms in a low Reynolds number regime near a no-slip wall has been a subject of growing interest in recent years because of its importance in many health and environmental problems. In addition to the changes in the swimming speed and energy expenditure of organisms in the presence of a wall, unexpected interesting swimming dynamics has been reported in recent experiments. In this study, the hydrodynamics of an archetypal low-Reynolds number swimmer, called "squirmers," near a wall has been numerically studied. Depending on the swimming mechanism and swimming direction, three different modes are distinguished: (a) squirmer escaping from the wall, (b) squirmer swimming along the wall keeping a constant height and orientation angle and (c) squirmer swimming near the wall in a periodic trajectory.

[1]This work is supported by NSF Grant No. CBET-1150348- CAREER.

1B.00007 Simulations of the burst and coast swimming behavior of fish[1] , QUAN ZHOU, Princeton University, KEITH MOORED, Lehigh University, ALEXANDER SMITS, Princeton University, Monash University — An investigation into the burst and coast swimming behavior of fish is simulated with a 2-D, inviscid Boundary Element Method. The fish is modeled as a thin pitching panel that is allowed to free swim. A simple drag model is used where drag is proportional to the velocity squared in order to calculate the cruising velocity. The burst-coast behavior is modeled by a coasting phase, where the panel is motionless, and a burst phase, where the panel pitches with a single sine wave motion. Varying the frequency of the fin-beat and the duration of the duty cycle (the ratio of the burst-phase to the entire period), it is found that it is possible to alter swimming motion to yield a decrease of 50% in the cost of transport with no sacrifice of time-averaged cruising velocity. The analyses of the wake structure demonstrate how vortices shed by the fish affect and shape swimming dynamics.

[1]Supported by the Office of Naval Research under Program Director Dr. Bob Brizzolara, MURI grant number N00014-08-1-0642.

1B.00008 Reduction of Urinary Tract Infections Caused By Urethral Catheter through the Implementation of Hydrophobic Coating and Geometrical Modifications , AYA GARE, Florida Atlantic University — Catheter-Associated Urinary Tract Infection (CAUTI) is the most common nosocomial infection in the U.S. healthcare system. The obstruction of urine caused by confined air bubbles result in the development of urinary back-flow and stagnation, wherein microbial pathogens could multiply rapidly and colonization within catheters become commonplace. Infections can be prevented by aseptic insertion and the maintenance of a closed drainage system, keeping high infection control standards, and preventing back-flow from the catheter bag. The goal of this study is to assess the effectiveness of a simple, low cost, modification that may be implemented into current catheter designs to reduce the incidence of CAUTI. Using the principle of transmission of fluid-pressure and the Young-Laplace equation for capillary pressure difference, this research focuses on improving the liquid flow in the presence of confined bubbles to prevent stagnation and reflux of bacteria-ridden urine into the body. Preliminary experiments are performed on a variety of tubes with hydrophobic-coating the interior, as well as geometrically modifying the tubes. Proof-of-Concept Prototype tubes are used to represent the drainage system of the catheter structure.

1B.00009 Dynamics of Surfactant Liquid Plugs at Bifurcating Lung Airway Models , HOSSEIN TAVANA, University of Akron — A surfactant liquid plug forms in the trachea during surfactant replacement therapy (SRT) of premature babies. Under air pressure, the plug propagates downstream and continuously divides into smaller daughter plugs at continuously branching lung airways. Propagating plugs deposit a thin film on airway walls to reduce surface tension and facilitate breathing. The effectiveness of SRT greatly depends on the final distribution of instilled surfactant within airways. To understand this process, we investigate dynamics of splitting of surfactant plugs in engineered bifurcating airway models. A liquid plug is instilled in the parent tube to propagate and split at the bifurcation. A split ratio, R, is defined as the ratio of daughter plug lengths in the top and bottom daughter airway tubes and studied as a function of the 3D orientation of airways and different flow conditions. For a given Capillary number (Ca), orienting airways farther away from a horizontal position reduced R due to the flow of a larger volume into the gravitationally favored daughter airway. At each orientation, R increased with $0.0005 < Ca < 0.05$. This effect diminished by decrease in airways diameter. This approach will help elucidate surfactant distribution in airways and develop effective SRT strategies.

1B.00010 Computational analysis of wake structure and body forces on marine animal research tag , MATTHEW ROSANIO, Syracuse University, JACOB MORRIDA[1], University of Notre Dame, MELISSA GREEN, Syracuse University — The Acousounde 3B marine animal research tag is used to study the relationship between the sounds made by whales and their behaviors, and ultimately to improve whale conservation efforts. In practical implementation, some researchers have attached external GPS Fastloc devices to the top surface of the tag, in order to accurately record the position of the whales throughout the deployment. There is a need to characterize the flow over the tag in order to better understand the body forces being exerted on it and how wake turbulence could affect noise measurements. The addition of the GPS Fastloc exacerbates both of these concerns, as it complicates the hydrodynamics of the device. Using CFD techniques, we were able to simulate the flow over the tag with a GPS attachment at multiple yaw angles. We used Pointwise to construct the mesh and Fluent to simulate the flow. We have also used flow visualization to experimentally validate our computational results. It was found that the GPS has a minimal effect on the wake of the tag at a 0 degree offset from the freestream flow. However, at increasing offset angles, the presence of the GPS greatly increased the amount of wake turbulence observed.

[1]Performed work while undergrad at Syracuse

1B.00011 Physics of Dielectrophoretic Trap by Analogy with Electrophoretic Paul Trap[1] , JAE HYUN PARK, Gyeongsang National University, South Korea — Dielectrophoresis (DEP) is defined as the motion of suspended particles in solvent resulting from polarization forces induced by an inhomogeneous electric field. DEP has been utilized for various biological applications such as trapping, sorting, separation of cells, viruses, nanoparticles, etc. The analysis of DEP trap has been so far based on the period-averaged pondermotive forces only while the dynamic features of DEP trapping have not been attracted. However, the recent study about aqueous electrophoretic Paul trap showed that a close relation between particle properties and their random motions, which cannot be understood via pondermotive effects. Similar to this, the present study reveals a detailed understanding of dynamic responses of DEP trap and their relation to various system parameters. The analogy with electrophoretic Paul trap is emphasized.

[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 (2012R1A1A1042920).

1B.00012 BUBBLES —

1B.00013 Saturation of the Afterbounce Shape Instability in Single Bubble Sonoluminescence; Theory and Experiment[1] , MOGENS LEVINSEN, Niels Bohr Institute, University of Copenhagen — Excitation of the afterbounce instability represents one route to bubble death in single-bubble sonoluminescence. By taking the existing first order theory for excitation of shape instabilities represented by expansion in a spherical harmonic to second order thereby mixing spherical harmonics of different orders, we show that the exponential growth into bubble disruption in a certain parameter regime is checked and a saturated stable state of shape distortion is possible. Experimental evidence provided by Mie scattering is presented and a possible connection to simultaneous spatially anisotropic light emission discussed.

[1]Work supported by The Danish Council for Independent Research; Natural Sciences.

1B.00014 COMPUTATIONAL FLUID DYNAMICS —

1B.00015 Large Eddy Simulation of a turbulent flow in two dimensional dunes using an immersed boundary method , GETNET AGEGNEHU, HEATHER D. SMITH, Louisiana State University — The flow over dunes separates at the crest, generating a shear layer which has a big role for energy dissipation and formation of coherent structures. Large Eddy Simulations using bodyfitted and immersed boundary grids are performed to study the detailed flow dynamics that occurs in a fixed two dimensional dunes. We used a three-dimensional, non-hydrostatic solver; OpenFOAM for this study. The immersed boundary method was implemented using a discrete forcing approach with direct imposition of the boundary conditions. A periodic boundary condition is imposed in both the streamwise and spanwise directions. No-slip and free slip conditions are applied for the bottom and top walls respectively. The flow is forced with a pressure gradient which yields the mean velocity. The numerical results have been quantitatively compared with an experimental data for the mean flow and turbulence profiles. Resolved streamwise velocity profiles from both the immersed boundary and bodyfitted grids are in a good agreement with the experimental data. A good correlation of turbulent intensities and instantaneous flow fields are also observed between the two methods. It is also shown that the numerical model overestimates the vertical velocity profiles in the leeward side of the dune.

1B.00016 A Brownian dynamics simulation of a colloidal particle in an alternating electric field very near an electrode , LEI WANG, DENNIS PRIEVE, Carnegie Mellon University — In previous experiments, a single 6 μm sphere, immersed in a 0.15 mol/m^3 electrolyte solution, was put in an alternating electric field (6 kV/m, 100 Hz to 10 kHz) acting normal to a nearby planar electrode. Even in the absence of the applied field, the particle is confined by a potential energy well formed by gravitational attraction and double-layer repulsion. While monitoring the elevation of the particle (order of 300 nm) with Total Internal Reflection Microscopy at millisecond intervals and with the AC field, the particle was observed to experience a steady attraction to the electrode, even when the deterministic oscillations were imperceptibly small. While dielectrophoresis could produce a steady attraction, the observed attraction has a frequency dependence which is not consistent with this force. In this work, we use Brownian dynamics simulation to explore the role of several nonlinearities in the equation of motion: 1) a position-dependent drag coefficient, 2) a position-dependent oscillating force and 3) a non-parabolic shape for the confining potential energy profile (non-linear spring).

1B.00017 CONVECTION AND BUOYANCY-DRIVEN FLOWS –

1B.00018 Rayleigh-Taylor instability: An initial condition study , TOM FINN, SARAT CHANDRA KUCHIBHATLA, DEVESH RANJAN, Texas A&M University — The Water Channel facility at Texas A&M University was employed to study the effects of initial conditions on Rayleigh-Taylor instability. Different stages of evolution of the unstable stratification of hot and cold water streams are experimentally recorded using planar imaging techniques and thermocouples. The Atwood number corresponding to a low temperature difference of 7-8 degrees C lies within the range of 0.001-0.002. Repeatable and controllable multimodal initial conditions of up to 11 modes are generated using a flapper mechanism. Dependence of flow behavior on initial wavelength and phase angle are deduced by using sets of experimental cases. Integral mixing width, molecular mixing between the water streams and fine scale mixing of scalars are studied using Planar Laser Imaging Fluorescence, (PLIF) technique. Dependence of these variables on initial condition and their behavior at late times is studied. Anisotropy in the flow field is currently being studied using Particle Image Velocimetry.

1B.00019 Two-Way Natural Convection of Divided Statically Unstable Fluid Layers Through Small Openings[1] , CHRISTIA TSAI, Florida Atlantic University — The diffusion and convection occurring between a liquid interfaces is studied extensively by many researchers, but the natural convection of unstable fluid layers through small openings has received little attention. The subject could be important in the study of leaking oil or oil release on the sea bed. The diffusion and convection across the liquid interface are studied using flow visualization techniques in conjunction with high speed photography to elucidate this particular fluid mechanics of natural convection occurring unstably. A two-way natural convection occurs in which vertical density stratification is exhibited on both top and bottom layers. In addition to the density stratification, a horizontal density gradient is formed, resulting in an internal wave near the bottom of the tank. Both single opening and multiple openings are investigated. The interactions between multiple openings are revealed. For future study, measurement and micro-beads will be added to the experiment for more detail observation. Two-way natural convection through small opening has potentials in many directions of study upon varies instabilities observed in this experiment.

[1]Tsung-Chow Su

1B.00020 DROPS –

1B.00021 Investigation of interfacial phenomena and thermocapillary effect on drop evaporation in reduced gravity condition , JINGCHANG XIE, HAI LIN, Institute of Mechanics, CAS — Based on ground-based experiments, a drop evaporation experiment will fly aboard Chinese recoverable satellite in the near future This experiment will focus on the interfacial phenomena of phase chance, heat and mass transfer and the effect of thermocapillary convection on drop evaporation process Close attention will also be paid to the contact angle behavior, the triple line shifting and their relations Our ground-based experiments observed the interior flow field and the gaseous exterior of small suspended evaporating drops, the temperature distributions inside and outside the drops. Both good heat conductor and heat insulating material were used as substrate materials to investigate their influence on heat transfer and surface temperature distribution of an evaporating drop Experimental results indicate that for a drop evaporating in ambient temperature without substrate heating, temperature gradients existed along the drop surface which results in stable thermocapillary convection and cells appeared near the surface throughout entire evaporating process. The thermocapillary convection greatly changed drop's interior temperature distribution and the way of energy and mass transfer. Temperature jump or discontinuity was also measured at drop free surface.

1B.00022 Size distribution of spray droplets at different temperature , ILDOO KIM, HYUNG JU LEE, HO JIN CHOI, KI-YOUNG HWANG, Agency for Defense Development, AIR-BREATHING ENGINE DIVISION TEAM — Atomization of a liquid jet through an injection nozzle is not only of fundamental interests but also crucial to many real-life applications like engines, ink-jet printers, flow cytometry. Because of practical importance, there have been many studies on the atomization mechanism such as its dependence on the nozzle shape, ambient air pressure and etc. In this study, we investigate the atomization characteristics focused on its dependence on fluid temperature. We varied the temperature of the fluid from -30 °C to 300 °C, and it is injected through a nozzle pneumatically. Such cold or hot jet of fluid is atomized in the flow-controlled chamber, and the size distribution of the spray droplets was measured by optical technique.

1B.00023 Droplet motion driven by electro-elasto-capillary effects , JAYMEEN SHAH, XIN YANG, YING SUN, Drexel University — The motion of droplets on natural and synthetic fibers underlines many technological applications including flexible displays, insulation, and smart filters. However, there is a lack of fundamental understanding of the coupled electrical, elastic, and capillary forces on droplets in fiber networks. In the present study, the motion of a water droplet suspended between two electrically insulated fibers of different Young's modulus, lengths and diameters are examined under electric fields. The results on rigid fibers reveal a critical voltage, under which the droplet remain stationary. Above this critical voltage, droplet self-propulsion is observed as a result of the interplay of electro, elasto and capillary forces on the droplet. The effects of the inter-fiber distance and Young's modulus on droplet motion are also discussed. The controllable motion of droplets can be used to manipulate or transport liquid at small scales.

1B.00024 Stable Drop Formation and Deposition Control in Ink Jet Printing of Polyvinylidene Fluoride Solution , NATHANIEL THORNE, XIN YANG, YING SUN, Drexel University, COMPLEX FLUIDS AND MULTIPHASE TRANSPORT LAB–DREXEL UNIVERSITY TEAM — Using inkjet printing as an additive fabrication method is an enabling technology for low-cost, high-throughput production of flexible electronics and photonics. Polymeric materials, such as Polyvinylidene fluoride (PVDF), are widely used as dielectric materials for microelectronics, batteries, among others. However, due to its large molecular weight and incompatibility with moisture in air, the stable drop formation of PVDF solution is quite challenging. In this study, we examine the effects of solute concentration, nozzle back pressure, ejection waveform, and ambient moisture on the formation of PVDF droplets. The deposition dynamics of inkjet-printed PVDF solutions are then examined as a function of the solvent concentration. Bi-solvents of different surface tensions and vapor pressures are used to induce Marangoni flows in order to suppress the coffee-ring effect. The deposition of a single droplet and the interactions between multiple drops are examined for a better control of the deposition uniformity. Printing of lines and patterns with reduced instability is also discussed.

1B.00025 Evolution of vapor into a Leidenfrost layer during drop impact , SANG JUN LEE, JI SAN LEE, NAMSEOP KWON, POSTECH, BYUNG MOOK WEON, Sungkyunkwan University, KAMEL FEZZAA, Argonne National Laboratotry, JUNG HO JE, POSTECH — When a liquid drop impacts a solid surface heated above the Leidenfrost temperature, the drop rebounds, known as Leidenfrost effect. This phenomenon plays an important role in many cooling and transfer processes involved in fuel combustion or spray cooling. In this study, we investigated the evolution of vapor into a Leidenfrost layer during drop impact using ultrafast X-ray phase-contrast imaging that allowed us to directly visualize the dynamic profiles of drop impact at 150 – 550C. Initially, we find that nucleation of vapor occurs during drop spreading, forming vapor bubbles. We then reveal that vapor bubbles collapse on the solid surface during drop recoiling, contributing to the formation of a Leidenfrost layer. Furthermore, we studied the effects of temperature and impact velocity on the bouncing dynamics of the drop.

1B.00026 FLOW CONTROL –

1B.00027 Investigation of Flow around Cylinder with Parallel Slit in a Circular Pipe using Flow Visualization Approach , ARUMURU VENUGOPAL, LAVISH ORDIA, AMIT AGRAWAL, S.V. PRABHU, Indian Institute of Technology Bombay — Flow visualization experiments behind a cylinder with parallel slit placed inside a circular pipe are carried out with water as the working medium. Dye injection technique is employed to visualize the complex vortex formation mechanism behind the bluff bodies. Various wake parameters like Strouhal Number, wake width and the Vortex formation length are calculated from the recorded images with the help of commercial software MATLAB. Three different slit widths with s/d values of 0.1, 0.2 and 0.4 were chosen to study the effect on vortex formation mechanism and the corresponding wake parameters. The dual body character is expected to creep in at higher slit widths. Vortex formation both from the outer and the inner surfaces is observed. Symmetric vortex formation from the outer surface is observed. A separation bubble from each of the inner surface is formed that detaches itself from the bluff body to form a vortex at higher Reynolds. The separation bubble is sensitive to disturbance which is observed in the changing biasness on either side which also results in the transition from symmetric to alternate primary vortices. Their interaction with the outer vortices is observed to effect the strength of the outer primary vortices.

1B.00028 GENERAL FLUID DYNAMICS –

1B.00029 Modulation of flow field due to near-bank roots in small rivers , CHRISTIAN FRIAS, JORGE ABAD, University of Pittsburgh, EDDY LANGENDOEN, U.S. Department of Agriculture — It is well known that the presence of vegetation or log jams decreases the bank shear stress exerted by the water on a river. This fact is used in river restoration to design bank erosion control structures such as engineered log jams or streambank revegetation zones. Also it has been observed in small rivers the presence of exposed near-bank roots and rootwads that could have a similar effect as riparian vegetation or log jams. However, the role of them on the modulation of river bank erosion and sediment transport is still not well understood. An exposed root in the river does not only modify the averaged shear stress or the averaged velocities on the flow field but it changes the instantaneous hydrodynamics too. Thus, it is expected that a root or rootwad produce turbulence coherent structures in the flow field. The analysis of these turbulence coherent structures will give a better insight of the relationship between rootwads geometry and flow field modulation because of them. Herein it is presented a Large Eddy Simulation (LES) of an exposed rootwad at selected small creek in Pennsylvania. The geometry of the rootwad was measured with a terrestrial LiDAR system. The results will be used to characterize the flow field turbulence and associate it to the bank erosion.

1B.00030 Performance test of a low cost roof-mounted wind turbine[1] , BERNARDO FIGUEROA-ESPINOZA, Universidad Nacional Autónoma de México, ROBERTO QUINTAL, Universidad Autónoma de Yucatán, CLÉMENT GOURIOU, Orleans Polytech Engineering School, ALICIA AGUILAR, Universidad Michoacana de San Nicolas de Hidalgo — A low cost wind turbine was implemented based on the ideas put forward by Hugh Piggot in his book "A wind turbine recipe book," where such device is developed using materials and manufacturing processes available (as much as possible) in developing countries or isolated communities. The wind turbine is to be mounted on a two stories building roof in a coastal zone of Mexico. The velocity profiles and turbulence intensities for typical wind conditions on top of the building roof were analyzed using numerical simulations (RANS) in order to locate the turbine hub above any recirculation and near the maximum average speed. The coefficient of performance is going to be evaluated experimentally by measuring the electrical power generation and wind characteristics that drive the wind turbine on the field. These experimental results will be applied on the improvement of the wind turbine design, as well as the validation of a numerical simulation model that couples the wind characteristics obtained through CFD with the Blade Element Method (BEM) and an electro-mechanical model of the turbine-shaft-generator ensemble.

[1]Special thanks to the Coordinación de Investigación Científica of the Universidad Michoacana de San Nicolás de Hidalgo for their support.

1B.00031 Flow past 2-D Hemispherical Rigid Canopies , MARIA-ISABEL CARNASCIALI, University of New Haven — The flow past a 2-dimensional rigid hemispherical shape is investigated using PIV. Flow field measurements and images were generated with the use of a Thermoflow® apparatus. Results of this study are compared to prior work (APS DFD 2012 Session E9.00003) which employed CFD to investigate the flow in the near wake of hemispherical parachutes. The various sized gaps/open areas were positioned at distinct locations. The work presented here is part of a larger research project to investigate flow fields in deceleration devices and parachutes. Understanding the pitch-stability of parachutes is essential for accurate design and implementation of these deceleration devices but they present a difficult system to analyze. The flexibility of the parachute fabric results in large variations in the parachute geometry leading to complex fluid-structure interactions. Such flow, combined with flow through gaps and open areas, has been postulated to shed alternating vortices causing pitching/oscillations of the canopy. The results presented provide some insight into which geometric features affect vortex shedding and may enable the redesign of the baseline parachute to minimize instabilities.

1B.00032 GEOPHYSICAL –

1B.00033 An experimental analog for the study of waving marine grass in tidal currents , JULIA LEE, RAVI SINGH, SHREYAS MANDRE, Brown University — Tidal currents passing through submerged vegetation mix the fluid and facilitate various environmental and ecological transport processes. This fluid-vegetation interaction, where the submerged grasses behave like those on the ground waving from wind, results from a shear instability of the surrounding flow. We devise a two-dimensional lab scale analog of the fluid-vegetation interaction using ABS plastic filaments immersed in a soap film to simulate the grass blades in a tidal flow. The array of filaments spontaneously waves in response to the flow of the soap film. Our experimental system makes direct flow measurement possible for a detailed comparison with theory.

1B.00034 GRANULAR FLOWS –

1B.00035 Measurement of self diffusion in a two-dimensional complex plasma , T.E. SHERIDAN, Ohio Northern University — Complex (dusty) plasma is an open, weakly-damped system of charged, microscopic particles which interact through a long-range screened Coulomb force. We have experimentally characterized diffusion in a two-dimensional (2d) liquid complex plasma. The 2d complex plasma is heated naturally by a surrounding three-dimensional toroidal dusty plasma gas. The measured dust velocity distribution functions are isotropic Maxwellians, giving a well-defined kinetic temperature T. The mean-square displacement is found to increase linearly with time, indicating normal diffusion. Measured diffusion coefficients increase approximately linearly with T. The effective collision rate is dominated by dust-dust collisions rather than neutral gas drag.

1B.00036 Contact Dynamics Models for Spacecraft-Regolith Interactions[1] , CHRISTINE HARTZELL, MELANY HUNT, California Institute of Technology — Granular mechanics simulations are currently used in the planetary science community in order to understand the evolution of asteroids, which are believed to be self-gravitating conglomerates of boulders and smaller grains. These simulations are typically done with Hard-Sphere or Soft-Sphere Discrete Element Method (DEM) codes. However, asteroids are increasingly being considered as exploration targets for the future spacecraft. Due to the very low gravity on the surface of these bodies (in some cases, six orders of magnitude less than Earth's gravity), it is not reasonable to design sample collection devices or mobility systems for future spacecraft based solely on Earth-based experimentation. However, there are limitations to using DEM codes for dense granular systems. Thus, we are creating a Contact Dynamics model to aid in the design of future spacecraft. In addition to its necessity for the design of spacecraft for asteroid exploration, granular mechanics simulations will also reduce the cost and risk of designing future mobility and sample collection systems for spacecraft heading to the Moon and Mars. We will present the current status of our Contact Dynamics code for monodisperse, spherical grains.

[1]Funding provided by Keck Institute for Space Studies

1B.00037 INDUSTRIAL APPLICATIONS –

1B.00038 Electric-Field-Enhanced Jumping-Droplet Condensation , NENAD MILJKOVIC, DANIEL PRESTON, Massachusetts Institute of Technology, RYAN ENRIGHT, Bell Labs Ireland, ALEXANDER LIMIA, EVELYN WANG, Massachusetts Institute of Technology — When condensed droplets coalesce on a superhydrophobic surface, the resulting droplet can jump due to the conversion of surface energy into kinetic energy. This frequent out-of-plane droplet jumping has the potential to enhance condensation heat and mass transfer. In this work, we demonstrated that these jumping droplets accumulate positive charge that can be used to further increase condensation heat transfer via electric fields. We studied droplet jumping dynamics on silanized nanostructured copper oxide surfaces. By characterizing the droplet trajectories under various applied external electric fields (0 – 50 V/cm), we show that condensation on superhydrophobic surfaces results in a buildup of negative surface charge (OH-) due to dissociated water ion adsorption on the superhydrophobic coating. Consequently, the opposite charge (H3O+) accumulates on the coalesced jumping droplet. Using this knowledge, we demonstrate electric-field-enhanced jumping droplet condensation whereby an external electric field opposes the droplet vapor flow entrainment towards the condensing surface to increase the droplet removal rate and overall surface heat transfer by 100% when compared to state-of-the-art dropwise condensing surfaces. This work not only shows significant condensation heat transfer enhancement through the passive charging of condensed droplets, but promises a low cost approach to increase efficiency for applications such as atmospheric water harvesting and dehumidification.

1B.00039 INSTABILITY –

1B.00040 Theoretical models for the stability of a liquid ring on a substrate[1] , JAVIER A. DIEZ, ALEJANDRO G. GONZÁLEZ, Instituto de Física Arroyo Seco (CIFICEN-CONICET), Universidad Nacional del Centro de la Provincia de Buenos Aires, Tandil, Argentina, LOU KONDIC, Department of Mathematical Sciences, New Jersey Institute of Technology, University Heights, Newark, NJ 07102 — A viscous incompressible fluid ring on a partially wetting substrate is studied within the framework of long-wave theory. We found that static equilibria are posible in the presence of contact angle hysteresis. Their linear stability is carried out by using a slip model. A quasi-static approximation is also implemented to analyze longer times. This latter approach takes into account the concomitant variation of the instantaneous growth rates of the modes responsible for either collapse of the ring into a single central drop or breakup into a number of droplets along the ring circumference. We compare the results of these models with those obtained from nonlinear numerical simulations based on a complementary disjoining pressure model. We find remarkably good agreement regarding the expected number of drops forming during the breakup process. (J. Fluid Mech. **718**, 246 (2013))

[1]Supported by CONICET-Argentina grant PIP 844/2011 (AGG, JAD), and by NSF grant CBET-1235710 (LK)

1B.00041 MICROFLUIDS –

1B.00042 Response of microfluidic fuel cells to secondary flows , MASSIMILIANO ROSSI, CHRISTIAN J. KÄHLER, Bundeswehr University Munich — Microfluidic or membraneless fuel cells (MFCs) are a recent class of miniaturized fuel cells (Ferrigno et al. 2002, Choban et al. 2004) composed by a microchannel in which a parallel laminar stream of two fluids, a fuel and an oxidant, is established. The fuel and oxidant remain in contact but do not mix due to the absence of turbulence. The simple architecture and the fact that no expensive proton exchange membranes are needed make this configuration technologically very attractive, however the efficiency especially in terms of fuel utilization is still too low to be competitive for practical applications. One limitation is given by the formation of depletion boundary layers at the electrodes that worsen the red-ox reactions. A way to reduce this problem is to use transversal secondary flows to stir the fluid streams and replenish the depletion layers. In this study, we intend to characterize the performance of MFC with curved channels in which the transversal secondary flows are present in the form of two counter-rotating vortices known as Dean vortices. The characterization will be completed by simultaneous measurements of the current intensity and of the flow velocity performed with 3D Astigmatic Particle Tracking Velocimetry.

1B.00043 Molecular dynamics simulation of dewetting of ultra-thin liquid film with artificial dry patches , SUSUMU KONO, ICHIRO UENO[1], Department of Mechanical Engineering, Faculty of Science and Technology, Tokyo University of Science — Large scale molecular dynamics simulations of dewetting of ultra-thin liquid films on a solid substrate are carried out. In case of thin film of nanometer-scale thickness, the liquid film is ruptured spontaneously which is called spinodal decomposition. The instability generates the dry patches in the film. The dewetting process begins from the several dry patches. Finally, the liquid film varies to droplets on the substrate. In the present study, we focus on the dry patch distribution in the liquid film, and investigate the depending of the initial distribution of the dry patches on the final stage of the nanometer-scale droplet formation. First, the liquid film composed by LJ fluids covered solid substrate with the preset dry patches. Then the dewetting behavior based on the artificial dry patches distribution is observed. As a result, it is found that there exists a sharp threshold of the initial radius of the artificial patch to realize the spontaneous rupture. This threshold depends on the initial film thickness, contact line curvature and LJ liquid-solid parameter. In comparing with usual dry patches distribution due to spinodal decomposition to artificial one, the final droplets formation also depends on initial dry patches distribution.

[1]Research Institute for Science and Technology (RIST), Tokyo University of Science

1B.00044 Local fluorescence in micro channels for particle counting[1] , MARIANA CENTENO SIERRA, MATHIEU HAUTEFEUILLE, CATALINA STERN, Facultad de Ciencias, UNAM — We produce local fluorescence in polydimethylxiloxane (PDMS) microchannels with a low power laser. This technique can be used to count either particles or cells in microflows. A CCD webcam is mounted on the objective of a microscope to visualize the flow. Particles obstruct the fluorescence as they pass by, allowing for a simple counting method that is software controlled. We present the experimental setup and preliminary results.

[1]We acknowledge support from the Physics Department of the National University of Mexico

1B.00045 Study on fabrication of scaffold using three-dimensional electrohydrodynamic ink-jet technique[1] , HAN SEO KO, SOO-HONG LEE, PIL-HO LEE, DAE-HOON KIM, CHIANG WEI YU, SANG WON LEE, Sungkyunkwan University — The EHD ink-jet technique uses the electrostatic force by applied voltage between a nozzle and an electrode to fabricate a three-dimensional scaffold by accumulating layers. In this study, a PLA (Polylactide) which is a polymer material was used to make the biodegradable scaffold. The experiment was performed by various inks with different solvent ratios because the layer thickness and width on the substrate are influenced by the ink properties such as the solvent ratio and boiling point. The cone-jet mode which looks cone-shaped on the meniscus was used for the EHD jetting by various stage velocities and solvent ratios of the PCL material. The micro-zoom lens and the LED lamp were used to visualize the jetting performance. The three-dimensional printing was completed by the movement of the stages using the Gentry structure. The optimum condition was selected for the fabrication of the scaffold after investigating the width of the pattern and the thickness of the multiple layers.

[1]This work supported by the Basic Science Research Program through the National Research Foundation of Korea (NRF) funded by the Korean government (MEST) (No. S-2011-0023457).

1B.00046 Effect of Inhomogeneous Flow on Micro-scale Biomedical Gas , S. SEN, National Institute of Aerospace and College of William & Mary, Hampton, VA 23666 — In this work the effect of a radially varying parallel equilibrium flow on the stability of the Rayleigh-Taylor mode is studied in a micro-scale confined gas in a biomedical system. It is shown that the parallel flow curvature can completely stabilize the mode. The flow curvature also has a robust effect on the radial structure of the mode. Possible implications of these findings are also discussed.

1B.00047 MULTIPHASE FLOWS —

1B.00048 Energy and Momentum Transport in Microfluidic with Shear-driven Flows , S. SEN, National Institute of Aerospace and College of William & Mary, Hampton, VA 23666 — Transport of the pressure-driven perturbations with a sharp parallel velocity shear is studied in a microfluidic. Studies show when the second derivative of the parallel velocity with respect to the radial coordinate is positive, the linear mode may become unstable and turbulent momentum transport increases. On the other hand, when the second derivative of the parallel velocity is negative, the linear mode is completely stabilised and turbulent momentum transport reduces. Possible implications of this results in biomedical industry will be discussed.

1B.00049 NANOFLUIDS — .

1B.00050 Using instability of nanometric liquid Cu films on SiO2 substrates to determine the underlying van der Waals potential[1] , ALEJANDRO G. GONZÁLEZ, JAVIER A. DIEZ, Instituto de Física Arroyo Seco (CIFICEN-CONICET), Universidad Nacional del Centro de la Provincia de Buenos Aires, YUEYING WU, Department of Materials Sciences and Engineering, University of Tennessee, Knoxville, JASON D. FOWLKES, Center for Nanophase Materials Sciences, Oak Ridge National Laboratory, PHILIP D. RACK, Department of Materials Sciences and Engineering, University of Tennessee, Knoxville, LOU KONDIC, Department of Mathematical Sciences, New Jersey Institute of Technology — We study the instability of nanometric Cu thin films on a SiO2 substrate. The metal is melted by means of laser pulses for some tens of nanoseconds. The free surface destabilizes during the liquid lifetime, leading to the formation of holes at first and to metal drops on the substrate in later stages. By analyzing the Fourier transforms of the SEM images obtained during the metal film evolution, we determine the emerging length scales for both early and late stages of the instability development. The results are analyzed within the framework of a long-wave hydrodynamic model, which introduces van der Waals forces by means of disjoining and conjoining pressures. These forces are characterized by a pair of exponents for the ratio $h/h*$, where h is the liquid thickness and h_* is a residual one. We find that the pair $(3, 2)$ provides a good agreement for the relationship of the wavelength with maximum growth rate, λ_m, while other typical pairs, such as $(4, 3)$ and $(9, 3)$ do not provide accurate description of the experimental data (Langmuir **29**, 9378 (2013)).

[1]Supported by CONICET-Argentina grant PIP 844/2011 (AGG, JAD), and by NSF grants CBET-1235651 (PDR) and CBET-1235710 (LK).

1B.00051 Lithography-free nanofluidic concentrator based on droplets-on-demand system[1] , MIAO YU, HONGBO ZHOU, SHUHUAI YAO, Hong Kong University of Science and Technology — Biomarkers are usually low-abundance proteins in biofluids and below detection limit of conventional biosensors. Nanofluidic concentration devices allow efficient biomolecules trapping by utilizing ion concentration polarization near nanochannels. However, once the electric field is turned off, the electrokinetic concentration plug cannot maintain its concentration status and starts to diffuse. In order to maintain the high concentration and extract the concentrated sample for further analysis, a good approach is to encapsulate these plugs into water-in-oil droplets. Here we developed a nanofluidic concentrator based on droplet-on-demand generator to encapsulate concentrated sample in nL droplets. The lithography-free nanochannels were patterned by thermal cracking on the surface of PS Petri-dish. The resulting nanochannel arrays were 30 nm in depth. In combination with microchannels on PDMS, the micro-nano hybrid chip was developed. We used FITC solution to demonstrate that the chip significantly increased the sample concentration for more than 100 folds within 5 minutes. By tuning the pulsed pressure imposed by the solenoid valve connected to the concentration channel, the system can generate a desired volume of droplet with a target sample concentration at a prescribed time.

[1]This work was supported by the Research Grants Council of Hong Kong under General Research Fund (Grant No. 621110)

1B.00052 NON-NEWTOWNIAN FLOWS —

1B.00053 Toward Generating Low-Friction Nanoengineered Surfaces with Liquid-Vapor Interfaces , CHU WANG, XIN YONG, LUCY ZHANG, Rensselaer Polytechnic Institute — Using molecular dynamics (MD), we investigate the importance of liquid-vapor interface topography in designing low friction nanoengineered superhydrophobic surfaces. Shear flow is simulated on patterned surfaces with cylindrical nanoholes and nanopillars. We devise an approach to generate entrapped bubbles with large protrusion angles in MD simulations, where the relationship between the effective slip length and bubble meniscus curvature is attained. We show that protruded bubbles can induce significant friction which hinders the slip characteristics produced on liquid-vapor interfaces. We also demonstrate that the continuity of the liquid-vapor interface greatly influence slip. Good quantitative agreements with previous simulations and analytical models on the asymptotic behavior of slip length with varying gas fraction are obtained. Our results show that we can adopt ideas from continuum scale analysis to design nanoengineered surfaces with large slip, with the caution of detailed interface dynamics at nanoscale.

1B.00054 Parameter effects on shear stress of Johnson–Segalman fluid in Poiseuille flows , XIANG QIU, School of Science, Shanghai Institute of Technology, Shanghai 201418, JIANPING LUO, School of Mechanical Engineering, Shanghai Institute of Technology, Shanghai 201418, YULU LIU, Shanghai Institute of Applied Mathematics and Mechanics, Shanghai University, Shanghai 200072 — Exact solutions of shear stress versus velocity gradient and the numerical solutions of streamwise velocity distribution in radial direction of a JohnsonSegalman fluid in a circular pipe are obtained. The effects of material parameters, Weissenberg number, ratio of viscosities and slip parameter, on shear stress and streamwise velocity have been considered to investigate the discontinuous velocity derivatives and stick-slip phenomenon at the wall. We find that there is a non-monotonic relationship between the shear stress and rate of shear for certain values of the material parameters and consequently, the velocity profile has discontinuous derivatives. Moreover, the variational range of material parameters is given for the appearance of a non-monotonic relationship between the shear stress and the rate of shear. Finally, we have shown the exact expression of critical pressure gradient and also have given the conditions where spurt phenomena occur.

1B.00055 ABSTRACT WITHDRAWN –

1B.00056 PARTICLE-LADEN FLOWS –

1B.00057 Pressure Driven Flow of Inhomogeneous Suspensions: Experiments and Theory , ASHWIN VAIDYA, Montclair State University, MEHRDAD MASSOUDI, National Energy Technology Lab, DOE, SIOBHAN SOLTAU, GIN SANCHEZ, JILLIAN VARNER, JOSEPH FIORDILINO, Montclair State University — This study is devoted to the experimental and theoretical investigation of the pipeline flow of low volume fraction suspensions. We derive our motivation from questions concerning the feasibility of pipeline transport of biomass. Our experimental observations, based on a table-top scale study indicate an unusual relationship between flow rate and pressure gradient which has not been observed in homogeneous systems. For our system, which consists of (2%-6% volume fraction) mixtures of mulch/coffee powder/crushed leaves in water, we find that for a certain range of pressure gradients, the flow rate in fact decays for increasing pressures. Based on a generalization of the Newtonian fluid model, we mathematically model our mixture by taking the system's bulk viscosity and being dependent upon the pressure gradient. The resulting expression for flow rate is fitted to experimental data showing a very good correlation. The results of this study provide the only example of a system where a pressure dependent viscosity is valid at low pressures. We also consider a single phase non-Newtonian model for this system where the effects of shear rate and normal stresses are incorporated.

1B.00058 Study of Local Profiles Relative to the Particle Surface in a Forced Particle-Laden Turbulent Flow[1] , OSCAR CASTRO, University of Delaware, ORLANDO AYALA, Old Dominion University, LIAN-PING WANG, University of Delaware, LIAN-PING WANG COLLABORATION, ORLANDO AYALA COLLABORATION — Turbulent flows laden with solid particles, liquid droplets, or air bubbles are relevant to many engineering applications and biological and environmental processes. 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 interaction of a solid particle with the carrier flow when its size is comparable or larger than the Kolmogorov scale. Here we address the interaction of finite-size particles with the carrier fluid turbulence using lattice-Boltzmann-based, particle-resolving simulations. Our recent results (Comput. & Math. with Applications, DOI: 10.1016/j.camwa.2013.04.001) on forced turbulence laden with non-sedimenting solid particles at a particle-to-fluid density ratio of 5, solid volume fraction of 0.102, and particle diameter to Kolmogorov length ratio of 8.05 reveal that the enhanced viscous dissipation is related to the local flow profiles near the particle surface. Here we repeat this simulation and present more accurate local profiles by averaging over time in addition to space. We will also analyze how such profiles change with the particle volume concentration and the particle size relative to the Kolmogorov scales.

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

1B.00059 Two-way Interactions in Particle-Laden Turbulent Channel Flow[1] , CHENG PENG, OSCAR CASTRO, University of Delaware, ORLANDO AYALA, Old Dominion University, LIAN-PING WANG, University of Delaware — Most previous studies of two-way interactions in particle-laden turbulent channel flows were performed using the point-particle approach. Here we present preliminary results on two-way coupling of finite-size particles with turbulence in a channel flow. The lattice Boltzmann approach is used to resolve both the channel flow and the disturbance flows around moving particles. Results of single-phase turbulent channel flows are first compared to published benchmark DNS results to validate the lattice Boltzmann approach. Preliminary results on turbulent particle-laden channel flow are analyzed at three levels: whole-field, phase-partitioned, and profiles as a function of distance from the surface of solid particles. We will examine the effects of finite particle size on the mechanisms of energy production and dissipation. Specifically, the two-way interactions near the channel wall are contrasted with those away from the walls. Results will be compared to those based on the point particle approach. We will also study how the results change with particle size, particle-to-fluid density ratio, and particle volume fraction.

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

1B.00060 Computationally and experimentally assessed gravity-driven, mono- and bidisperse, particle-laden flows , SHREYAS KUMAR, Harvey Mudd College, KAIWEN HUANG, UCLA, MATT HIN, Cornell University, GILBERTO URDANETA, ALIKI MAVROMOUSTAKI, JEFFREY WONG, UCLA, SUNGYON LEE, Texas A&M, ANDREA BERTOZZI, UCLA — We present an experimental study which investigates the motion of granular materials in mono- and bi-disperse suspensions consisting of silicone oil, glass and ceramic beads. The beads are of distinct densities both denser than the oil but of approximately the same size. A finite volume of slurry is allowed to flow down an inclined plane and the subsequent flow development is recorded. The system parameters are the angle of inclination, the total particle concentration and the relative amount of heavy (ceramic) to light (glass) beads. Similarly to the results observed in previous studies of mono-disperse slurry flows, in bidisperse suspensions, there exist two stable flow regimes: the first one involves settling of particles to the substrate while, in the second one, the particles aggregate at the front of the flow. We carry out a series of experiments to investigate the effect of system parameters on the resulting flow regime patterns and compare our results with a theoretical model which incorporates the effects of shear-induced migration and sedimentation in bidisperse suspensions of negatively buoyant particles. Further, we use fluorescent particle beads to compare the particle spatio-temporal evolution observed in experiments against numerical simulations.

1B.00061 ABSTRACT WITHDRAWN –

1B.00062 POROUS MEDIA FLOWS –

1B.00063 ABSTRACT WITHDRAWN –

1B.00064 ROTATING FLOWS –

1B.00065 The interfacial dynamics between two immiscible rotating fluids , HUA-YI (MAGGIE) HSU, Department of Mechanical Engineering, National Taipei University of Technology, NATIONAL SCIENCE COUNCIL, TAIWAN COLLABORATION — We numerically investigated the topological interface change occurring between two rotating, immiscible, stratified fluids. We simulate the two- layer fluid in a cylindrical container which is driven by a disk with a constant angular velocity. The upper fluid is of higher viscosity than the lower one, and the ratio of the radius of cylindrical container and the depth of the upper fluid is set to be one of the parameters. The surface tension between 2 fluids is one of the key control factors which change the topological interface. The interface behaviors were found over a wide range of parameters. The topological interface shape will be found such as: hill, plateau, bell, drop formation, and chaos. We also investigate the size of drop using different parameters.

1B.00066 SUSPENSIONS —

1B.00067 Liquid flow between hydrodynamicaly interacting particles in confined systems , ALVARO GOMEZ MARIN, MASSIMILIANO ROSSI, CHRISTIAN J. KAEHLER, Bundeswehr University Munich — Particle suspensions in confined geometries are greatly complex systems since they introduce a high degree of complexity into the otherwise linear Stokes flows. Very recently, new mechanisms of instability have been identified in simulations in confined shear-flows of non-Brownian dilute particle solutions (Zurita-Gotor et al., J. Fluid Mech. 592, 2007, and Phys. Rev. Lett. 108, 2012). In this study we will focused on particle pairs interacting with walls, which requires the use of micro-confined systems. By the use of Astigmatism-PTV on particle solutions with different fluorescent characteristics, we will solve both the non-Brownian interacting particle trajectories and the flow around them in order to elucidate the details of the hydrodynamic particle-particle interactions.

1B.00068 Measurements of wall shear stress in a planar turbulent Couette flow with porous walls , PAUL BEUTHER, Kimberly-Clark Corp. — Measurements of drag on a moving web in a multi-span festoon show a stronger than expected dependency on the porosity of the web. The experiments suggest a wall shear stress 3-4 times larger than non-porous webs or historical Couette flow data for solid walls. Previous DNS studies by Jimenez et.al. (JFM Vol 442) of boundary layers with passive porous surfaces predict a much smaller increase in wall shear stress for a porous wall of only 40%. Other DNS studies by Quadrio et.al. (JFM Vol 576) of porous walls with periodic transpiration do show a large increase in drag under certain periodic conditions of modest amplitude. Although those results are aligned in magnitude with this study, the exact reason for the observed high drag for porous webs in this present study is not understood because there was no external disturbance applied to the web. It can be hypothesized that natural flutter of the web results in a similar mechanism shown in the periodic DNS study, but when the natural flutter was reduced by increasing web tension, there was only a small decrease of the drag. A key difference in this study is that because of the multiple parallel spans in a festoon, any transpiration in one layer must act in the opposite manner on the adjacent span.

1B.00069 Atmosphere-ocean exchanges over slow and fast wave fields , QI LI, ELIE BOU-ZEID, Department of Civil and Environmental Engineering, Princeton University, Princeton, NJ, NIKKI VERCAUTEREN, Department of Physical Geography and Quarternary Geology, Stockholm University, Stockholm, Sweden, MARC PARLANGE, School of Architecture, Civil, and Environmental Engineering, École Polytechnique Fédérale de Lausanne, Lausanne, Switzerland — We explore the influence of surface gravity waves on momentum and scalar exchanges between the atmosphere and underlying water surfaces, based on field experimental data sets. The existence of both slow (occurring under active local wind forcing) and fast (swell) waves and their interactions with the atmosphere show unique features compared to flow over fixed walls. While turbulence and fluxes over slow waves surfaces share many features with flow over fixed surfaces, fast moving waves complicate the picture with different ranges of scales transporting momentum in opposite vertical directions. We further show that, especially under fast-wave conditions, the surface waves' phase velocity and directionality influence the form drag at the surface, inducing a misalignment between the surface stress and wind velocity vectors. This improved understanding allows us to re-approach how classic loglaws and their extension to non-neutral conditions (the Monin-Obukhov similarity theory) are formulated and applied over the marine/water surface. Particularly, we show that the appropriate air velocity parameter to use in these laws is the difference between the wind velocity component parallel to the wave propagation direction and the wave phase velocity.

1B.00070 Comparison between Prandtl, Navier-Stokes and Euler solutions for 2D flows in the presence of solid boundaries , MARIE FARGE, LMD-CNRS-IPSL, Ecole Normale Superieure, Paris, France, ROMAIN NGUYEN VAN YEN, MATTHIAS WAIDMANN, Fachbereich Mathematik, Freie Universitaet, Berlin, Germany, KAI SCHNEIDER, M2P2-CNRS and Faculte des Sciences, Aix-Marseille Universite, France, RUPERT KLEIN, Fachbereich Mathematik, Freie Universitaet, Berlin, Germany — In 1904 Prandtl introduced the notion of boundary layer (BL), assuming all viscous energy dissipation takes place only in the BL (as long as it remains in contact with the body) whose thickness is inversely proportional to the Reynolds number, Re. He derived the BL equation and succeeded to asymptotically match its solution with that of an inviscid fluid flow governed by Euler's equation outside the BL. In the poster we address the following question: does energy dissipate when the BL detaches from the solid body? We consider a jet, modeled as a vorticity dipole, impinging onto a wall, that we study by Direct Numerical Simulation to see how solutions behave in the vanishing viscosity limit (equivalent to the limit of large Re). Starting from the same initial flow and the same geometry, we compare the solutions obtained for Euler's equation, Prandtl's equation, and Navier-Stokes equation, using different numerical methods. We observe that in the vanishing viscosity limit energy dissipation does not tend to zero, in a BL whose thickness scales as $Re^{-1/2}$ (as predicted by Prandtl's 1904 theory), but produces vortices at the wall which entrain the BL and roll it up to form a dissipative structure, whose thickness scales as Re^{-1} (Kato, 1984), which detaches from the wall.

1B.00071 Non-Dimensionalization and Scaling of Helmholtz Equation and Schrodinger Equation, Which Reformulated for Fluid Dynamics , AHMAD REZA ESTAKHR, Independent Researcher — In fluid mechanics, the Reynolds number (Re) is a dimensionless number $R_e = \frac{F_int}{F_vis}$ I defined Reynolds number in a different situation, through the Helmholtz equation which represents the time-independent wave equation, $\nabla^2\psi + k^2\psi = 0$ Now i consider wave vector k as Reynolds number per a characteristic linear dimension so, $\nabla^2\psi + \frac{R_e^2}{L^2}\psi = 0$ which led to the non-dimensionalization and scaling of Helmholtz equation, $L^2\nabla^2\psi + R_e^2\psi = 0$ this equation is applicable to fluid dynamics. then I reformulate schrodinger equation, $-\frac{\mu^2}{2\rho}\nabla^2\psi + U_v\psi = E_v\psi$ where the μ denotes viscosity, ρ is density, U_v and E_v are potential and total energy per unit volume. non-dimensionalization tricks: $\frac{\mu^2}{2\rho} = \frac{\mu L v}{2R_e}$ where the v is velocity, L is linear dimension. now if we take factor of $\frac{\mu v}{L}$ from both side of equation, the Non-dimensionalization and scaling of schrodinger equation for fluid dynamics will be, $-\frac{L^2}{2R_e}\nabla^2\psi + U_v^*\psi = E_v^*\psi$

1B.00072 Kinetic Energy-Preserving Discretization Schemes for High Reynolds Number Propulsive Applications[1] , AYABOE EDOH, ANN KARAGOZIAN, University of California, Los Angeles — The overarching goal of this project is to explore numerical approaches for the study of turbulent flows and to use them to explore the fundamental physics of combustion processes relevant to airbreathing and rocket propulsion systems. The present studies involve an investigation of kinetic energy-preserving discretization schemes that enable multiple ways for tracking acoustic versus particle waves in a compressible flowfield. Semi-discretized schemes have been explored, with a comparison of collocated and staggered grids and alternative multi-stage ODE schemes for time integration. The energy behavior of Crank-Nicolson collocated schemes is similarly explored, for example, for the scalar, inviscid Burger's equation and for the 1D compressible Euler equations. Comparisons of accuracy for different order codes with different dissipation coefficients and using different types of boundary conditions are made, including schemes that demonstrate the lack of requirement for artificial dissipation and strong energy preservation.

[1]Supported by the US Air Force and the UC Cota-Robles Fellowship program.

1B.00073 VORTEX DYNAMICS —

1B.00074 The Moore Singularity in the Evolution of a Vortex Sheet through Longitudinal Diffusion , UJJAYAN PAUL[1], Jawaharlal Nehru Centre for Advanced Scientific Research — The phenomenon of singularity in vortex sheets starting from analytic initial data has been studied in detail by various authors. Standard numerical solution of the vortex sheet using line vortex approximation indicates a singularity at a finite critical time.However, the possibility of a weak solution to the vortex sheet problem at all time has been raised before.The weak solution is based on the convergence of the vortex blob method in the limit of zero blob size. Regularization techniques have been applied on point vortex models. In a real fluid the problem of finite time singularity is eliminated by viscosity. Here we shall discuss and compare two possible regularization techniques applied on a vortex sheet model, which allows us to continue the evolution much beyond the critical time as predicted by Moore.

[1]Advisor: Roddam Narasimha

1B.00075 POST-DEADLINE —

1B.00076 Interfacial waves generated by contact line motion through electrowetting , JONGHYUN HA, JAEBUM PARK, Seoul National University, YUNHEE KIM, JUNGMOK BAE, Samsung Advanced Institute of Technology, HO-YOUNG KIM, Seoul National University — The contact angle of a liquid-fluid interface can be effectively modulated by EWOD (electrowetting on dielectric). Rapid movement of the contact line, which can be achieved by swift change of voltages at the electrodes, can give rise to interfacial waves under the strong influence of surface tension. Many optofluidic devices employing EWOD actuation, such as lenses, three-dimensional displays and laser radar, use two different liquids in a single cell, implying that the motions of the two liquids should be considered simultaneously to solve the dynamics of interfacial waves. Furthermore, the capillary waves excited by moving contact lines, which inherently involve slipping flows at solid boundaries, pose an interesting problem that has not been treated so far. We perform a perturbation analysis for this novel wave system to find the dispersion relation that relates the wavenumber, and the decay length over which the wave is dissipated by viscous effects. We experimentally corroborate our theory.

1B.00077 Eigenmode analysis of advective-diffusive transport in micromixers by the diffusive mapping method , PATRICK ANDERSON, MICHEL SPEETJENS, OLEKSANDR GORODETSKYI, TU Eindhoven, MAX GIONA, Sapienza, MIXING COLLABORATION — Advective-diffusive transport in microflows is studied by means of the diffusive map- ping method, a recent extension of the mapping method by Gorodetskyi et al. (Phys. Fluids 24, 2012) that includes molecular diffusion. This greatly expands the application area of the mapping technique and makes the powerful concepts of eigenmode decompo- sition and spectral analysis of scalar transport accessible to an important class of flows: inline micromixers with diffusion. The staggered herringbone micro-mixer is adopted as a prototypical three-dimensional micro mixer. Simulations with the diffusive mapping method are in close agreement with experimental observations in literature and expose a strong impact of diffusion on the transport. Diffusion enables crossing of Lagrangian trans- port barriers and thus smoothens concentration gradients and accelerates homogenization. Spectral analysis of the mapping matrix reveals this already occurs on a modal level in that individual eigenmodes progressively smoothen and spread out across transport bar- riers with stronger diffusion. Concurrently, the corresponding eigenvalues diminish and thus fundamentally alter the mixing process by invariably causing homogenization, irre- spective of the Lagrangian flow structure. This happens faster and exhibits an earlier emergence of the dominant eigenmode the stronger the diffusion. Lagrangian structures may still affect the spectral properties in that flows comprising both islands and chaotic seas typically result in a richer set of eigenmodes compared to cases with global chaos.

1B.00078 A combined RANS-LES simulation of a turbulent round jet in a large enclosure , SASAN SALKHORDEH, SAGNIK MAZUMDAR, D. TYLER LANDFRIED, University of Pittsburgh, ANIRBAN JANA, Pittsburgh Supercomputing Center, Carnegie Mellon University, MARK KIMBER, University of Pittsburgh — A combined RANS-LES simulation of a round turbulent jet confined in a large cylindrical enclosure is conducted. As the computational cost of doing LES over the entirety of the large enclosure is high, LES is done only near the jet axis. First, steady, axisymmetric RANS simulation of the confined jet is performed using a thin wedge-shaped slice of the enclosure. The RANS results are validated with experimental data from literature. LES is then performed in a small cylindrical domain around the jet, with initial and boundary conditions provided by the validated RANS results. After comparing six Sub-Filter Stress (SFS) models, the SFS model chosen for the LES simulation is a variant of the dynamic Smagorinsky model. The effect of inlet flow profile and turbulent fluctuations on the evolution of the jet is investigated. The influence of filter characteristics on simulation results is also analyzed. Finally, the LES results are compared with experimental measurements.

1B.00079 Dynamic behavior of electrowetting-based liquid prisms , JAEBUM PARK, JONGHYUN HA, Seoul National University, KYUHWAN CHOI, JUNGMOK BAE, HO-YOUNG KIM, Samsung Advanced Institute of Technology — A liquid prism is an optofluidic device consisting of two immiscible liquids, whose interface acts as a refractive surface. To steer a light beam that constructs optical images, the interface profile, or the contact angle, is modulated via electrowetting on dielectric (EWOD). Alternating current (AC) voltages are used for liquid prisms to stably maintain a desired contact angle without charge saturation in general. However, minute oscillations at the contact line are observed due to rapid changes of voltages under AC conditions, which may propagate into the interface leading to the deterioration of the optical quality. Here we find that the oscillation behavior is strongly correlated with the type of electrolytes, so that the solutions of small ions are more vulnerable to oscillations. We give an empirical relationship of the oscillation amplitude to the AC frequency, and theoretically analyze the salient features of the electrowetting-driven interface motion.

1B.00080 Space-Time Pressure-Velocity correlations in a high Reynolds number Turbulent boundary Layer , YOSHITSUGU NAKA, MICHEL STANISLAS, JEAN MARC FOUCAUT, SÉBASTIEN COUDERT, LML UMR CNRS 8107 — In the present study, we developed an experimental setup for the simultaneous measurements of the fluctuating pressure and the three velocity components in a high Reynolds number turbulent boundary layer. A quantitative measure of the extension of the space-time pressure-velocity correlations is given based on their reconstructed 3D distributions. The correlations between the fluctuating pressure at the wall and in the field and each three velocity components exhibit characteristic behavior. The pu correlation show significant Reynolds number dependence which is less evident for the two other correlations. The wall pressure velocity correlation are significantly different from the field ones. All correlations evidence strong relation of the pressure fluctuations with large scale coherent structures. These relations are significantly different for positive and negative pressure fluctuations at the wall.

1B.00081 Shear Driven-Streaming Potential Flow in a Charged Slit Microchannel , BEHNAM KHORSHIDI, SUBIR BHATTACHARJEE, Department of Mechanical Engineering, University of Alberta, Edmonton, Alberta T6G 2G8, Canada — The flow behavior in microfluidic devices is of great importance due to the need of precise control of the mass and momentum transport in these small scale channels. In the case of two-phase flow, e.g. the stratified flow of an oil layer above an aqueous phase, the situation becomes more interesting, but complicated. In most cases, the interface between the liquids is electrically charged due to the presence of the dissolved ions or colloidal particles. Therefore, there is a possibility that the physicochemical properties of the interface affect the flow behavior. The objective of the present study is to develop a fundamental understanding of flowing oil-water interface, with particular focus on the role of electrical forces acting at this layer. Analytical expressions are derived to describe the electrokinetic effects of electric double layer (EDL) on the shear-driven flow of an aqueous electrolyte solution between a moving and a stationary wall, the moving wall representing the charged oil-water interface. The flow field is obtained under a wide range of operating conditions. The results show that the velocity profile changes significantly depending on the surface potential of the moving wall, which reveals the importance of convective transport of ions near the mobile interface.

1B.00082 Experimental investigation and analysis of continuous flow through trace gas preconcentrator , JIHYUN KIM, University of Southern California — It was proposed by Muntz et al. in 2004 to study a micro/mesoscale continuous flow through trace gas preconcentrator, which could avoid the time fidelity problem. The preconcentrator for rarefied trace gas analysis, which is one part of a gas detector or analyzer, has been built and consists of a main flow channel, pumping chambers, and separation membranes that are located upper and lower surface of the main flow channel. The preconcentration is not from stop, adsorption, and release, but is caused by the gradually decreasing cross section of the main flow channel until release through the detection unit such as gas chromatography, mass spectrometry, or optical diagnostics. This has the possibility of achieving concentration increase of various gases in a carrier gas by using relatively simple micro/macroscale mass diffusion separation stages, and is suitable for improving the time accuracy of analytical systems. A series of experiments were conducted in an attempt to validate the available numerical data, such as the concentration and gas flow speed of the newly continuous preconcentration technology. This study involved experimental investigations to obtain a base-line comparison of the existing numerical predictions provided by the prototype preconcentrator.

1B.00083 Theoretical models for trace gas preconcentrators , JIHYUN KIM, University of Southern California — Muntz et al., in 2004 and 2011, had attempted to describe theoretical models about the shape of a main flow channel and the concentration ratio of trace gas for a Continuous Flow-Through Trace Gas Preconcentrator by concepts of net flux and mass flow rate respectively. The possibilities were suggested to obtain theoretical models for the preconcentrator even through they were not satisfied with experimental results, because the theoretical models were only considered for free molecular flow. In this study, new theoretical models based on net flux and mass flow rate have been applied for each regime; free molecular flow, transition flow, and hydrodynamic flow. There are comprehensive numerical models to describe entire regimes with the new theoretical models induced by mass flow rate, but the new theoretical models induced by net flux can be only obtained for the hydrodynamic flow. The numerical predictions were compared with existing experimental results of the prototype of the preconcentrator. The numerical predictions of hydrodynamic and transition flows by mass flow rate were close to the experimental results, but other cases were different to the experimental data. Nevertheless, the theoretical models can provide the possibility to develop the theory of preconcentrator.

1B.00084 Energy-Efficient Rate-Based Particle Separation[1] , DIANA LIEVANO, TATHAGATA BHATTACHARYA, JOSEPH MCCARTHY, University of Pittsburgh — The effective separation of particles is key to numerous processes and industries handling solid materials. Despite this fact, particle separations techniques remain typically quite "low tech" and often are energy-intensive (e.g., sieving) or environmentally unfriendly (e.g., froth floatation) or both. Rate-based separation processes, on the other hand, represent a unique approach to particle separation that has the potential to be more flexible, more efficient, and more environmentally friendly than existing "low tech" techniques. In the present paper, we highlight passive granular ratchets, where particles of differing properties flow through a device often called a Galton board. In this type of device, the gravity-driven flow of particles down an inclined plane causes collisions between the particles and the evenly distributed pegs along the board. Dissipative collisions between particles as well as between paticles and pegs results in a diffusion-like motion of particles perpendicular to the flow. The extent of separation (i.e., how far one type of particle is removed from another) depends on the different distances traversed by the two types of particles and, ultimately, on the collision rate and energy dissipation for particle-peg events. A simple theory, will be set.

[1]University of Pittsburgh

1B.00085 3D CFD Simulation of Horizontal Spin Casting of High Speed Steel Roll[1] , KONSTANTIN REDKIN, Swanson School of Engineering, Mechanical Engineering and Materials Science Department, University of Pittsburgh, Pittsburgh, PA, USA, BORIS BALAKIN, Department of Physics and Technology, University of Bergen, CHRISTOPHER HRIZO, WHEMCO Inc., Pittsburgh, PA, USA, JEFFREY VIPPERMAN, ISAAC GARCIA, University of Pittsburgh, UNIVERSITY OF PITTSBURGH TEAM, WHEMCO COLLABORATION, UNIVERSITY OF BERGEN COLLABORATION — The present paper reports some preliminary results on the multiphase modeling of the melt behavior in the horizontal spinning chamber. Three-dimensional (3D) computational fluid dynamics (CFD) model of the high speed steel (HSS) melt was developed in a novel way on the base of volume-of-fluid technique. Preliminary 3D CFD of the horizontal centrifugal casting process showed that local turbulences can take place depending on the geometrical features of the "feeding" arm (inlet), its position relative to the chamber, pouring rates and temperatures. The distribution of the melt inside the mold is directly related to the melt properties (viscosity and diffusivity), which depend on the temperature and alloy composition. The predicted liquid properties, used in the modeling, are based on actual chemical composition analysis performed on different heats.

[1]Acknowledgement of WHEMCO and United Rolls Inc. for supporting the program. Special appreciation for Kevin Marsden.

1B.00086 The effect of stress-free shapes on the red blood cell dynamics[1] , PROSENJIT BAGCHI, DANIEL CORDASCO, ALIREZA YAZDANI, Rutgers University — We present 3D numerical simulations on the effect of the two different stress-free shapes on the dynamics of red blood cells. We observe that in a high viscosity medium, the cell with a nearly-spherical stress-free state undergoes transition from tank-treading to tumbling at a much lower capillary number than the cell with a biconcave stress-free shape. The cell with the biconcave stress-free shape easily loses the biconcave shape and exhibits large time-periodic shape oscillation and membrane folding, while the cell with the nearly-spherical stress-free state retains the biconcave shape without any membrane folding. In a low viscosity medium, however, both stress-free shapes exhibit almost the same dynamics that is characterized by cell tumbling. We then compare the orbital reorientation of the cell for the two stress-free states. In the high viscosity medium, both cells undergo a precession motion orienting their symmetry axis towards the vorticity axis at low capillary numbers, or a kayaking motion orienting the axis towards the shear plane at higher capillary numbers. The capillary number for the precession-to-kayaking transition is observed to be higher for the biconcave stress-free shape than that for the nearly-spherical stress-free shape. At low viscosity medium, both shapes exhibit qualitative similar precession dynamics.

[1]Funded by a grant from National Science Foundation

1B.00087 Physical Prototype Development for the Real-Time Detection and Mitigation of Hazardous Releases into a Flow System[1] , SARA RIMER, NIKOLAOS KATOPODES, Univ of Michigan - Ann Arbor — The threat of accidental or deliberate toxic chemicals released into public spaces is a significant concern to public safety. The real-time detection and mitigation of such hazardous contaminants has the potential to minimize harm and save lives. In this study, we demonstrate the feasibility of feedback control of a hazardous contaminant by means of a laboratory-scale physical prototype integrated with a previously-developed robust predictive control numerical model. The physical prototype is designed to imitate a public space characterized by a long conduit with an ambient flow (e.g. airport terminal). Unidirectional air flows through a 24-foot long duct. The "contaminant" plume of propylene glycol smoke is released into the duct. Camera sensors are used to visually measure concentration of the plume. A pneumatic system is utilized to localize the contaminant via air curtains, and draw it out via vacuum nozzles. The control prescribed to the pneumatic system is based on the numerical model.

[1]NSF-CMMI 0856438

1B.00088 Cooling of a Tapped Granular Column[1] , ANTHONY ROSATO, LUO ZUO, New Jersey Institute of Technology - Granular Science Laboratory, DENIS BLACKMORE, New Jersey Institute of Technology - Mathematical Sciences Dept. — We present the results of a discrete element investigation of the cooling of a tapped column of uniform, inelastic spherical particles (d) as it evolves to a state of zero kinetic energy. A linear loading-unloading soft contact model is employed, while tapping is simulated by applying a half-sine pulse of amplitude a/d and frequency f to a rigid floor supporting the column. For sufficiently energetic taps, the column dilates and then contracts over a time scale t_s, which depends on the number of particles N, restitution coefficient e, as well as tap parameters (a/d, f). Simulation data for ($1 \leq N \leq 50$) with other parameters being held constant suggested that a time-averaged collision frequency f_c scaled with N. Values of t_s, determined by identifying the instant when the kinetic energy thereafter remained less than 0.001% of its maximum value, were well-correlated with the form $\alpha(e)N^{-1} + \beta(e)$. Lastly, simulations were in good agreement with physical considerations, suggesting that t_s should scale with $(1 - e^2)^{-1}$ and inversely with f_c.

[1]Supported in part by NSF Grant CMMI- 1029809

1B.00089 A Dynamical Systems Approach to the Alpha Problem for Rayleigh-Taylor , DANIEL ISRAEL, Los Alamos National Laboratory — Turbulent mixing of the unstable Rayleigh-Taylor layer is observed to exhibit self similar growth which scales as $h = \alpha A t g t^2$. This quadratic growth can be theoretically derived through several different approaches including bubble dynamics, flux balances (Cook et al., 2004), similarity theory (Ristorcelli and Clark, 2004), or simple turbulence modeling. In all these approaches, however, the value of α must be determined empirically. Furthermore, it is not clear from the theory whether α is universal. In fact, reported experimental values for α exhibit a wide variation, almost all of which are well above the values seen in simulations, as documented by Dimonte et al. (2004). That study concluded that all the variation could likely be explained by the presence, or absence, of long wavelength perturbations which can effect the growth for quite a long time. The current work provides a new tool for investigating the transient behavior. Starting with an advanced moment closure model and applying an integral method is shown to result in a set of ordinary differential equations which can be viewed as a low-order model of the turbulence as it evolves towards a self-similar state. Applying the tools of dynamical systems we can examine the possible trajectories of the system in state space. This suggests a new physical picture of how long wavelengths might create the appearance of a high value of α. It also gives us a new set of metrics for validating turbulence models for non self-similar problems.

1B.00090 Modeling Dilute Gas-Solid Turbulent Flows using Moment Methods , DENNIS DUNN, KYLE SQUIRES, None — Numerically modeling particle-fluid interactions in turbulent two-phase flows has proven quite useful, and although Lagrangian particle-tracking methods are a plausibly accurate approach, these models are often limited to dilute flows and can be inaccurate in regions of locally large particle concentrations where inter-particle interactions and effects of two-way coupling can be significant. These and other considerations motivate the current effort aimed at implementing Eulerian-based approaches that treat the particle phase as a continuum. The specific focus of the current effort is on modeling dilute particle-laden turbulence in which the gas-phase carrier flow is populated with a second phase of small, dispersed solid particles possessing material densities much larger than that of the carrier flow, and consequently large particle Stokes numbers. The approach adopted in this work is derived from the quadrature-based method of moments. Simulations are conducted of a particle-laden turbulent boundary layer. The gas-phase carrier flow is computed using DNS and the results show that the carrier flow drives the particulate phase via the drag force and with resulting structural interactions, e.g., preferential concentration of particles, similar to those observed in Lagrangian particle tracking simulations. Further comparisons are made against simulations using Lagrangian particle tracking of the dispersed phase and demonstrate the utility of the Eulerian approach, e.g., with statistical descriptors in reasonable agreement between the two methods.

1B.00091 The Effects of a Spatially Variant Velocity Field On Stretching: Intuitive Measures , JASON NIXON, DAVID BIGIO, None — Laminar mixing theory describes the 'goodness' of mixing as a function of increased surface area shared between two fluid species. Previous work describes mixing from a post processing perspective, as a function of stretching history, while disregarding the underlying flow. In this work, mathematical measures are derived which predict fluid-fluid interface behavior in a flow and explore the underlying flow field. This family of measures creates an intuitive basis for the exploration interfacial growth. One set of measure relates velocity to the principal directions while the second set relates interfacial orientation with the principal directions. To explore the usefulness of these new measures, they are simulated in three flows; shear flow, divergent flow, and the spatially variant lid driven cavity. In these geometries, the new family of measures proves valuable for demonstrating growth regime characteristics, transitions in growth regime, as well as other flow characteristics unique to each field. It has been shown that the changes in mixing consistent with reorientation occur after a rapid change in the relationship of the flow and principle directions. The second set of measures in this family allows for interface growth and modeled to be studied more objectively.

1B.00092 Nature-Inspired Airfoils for Environmental Noise Reduction , SUYEONG HAN, Cheong Shim International Academy, RICHARD KYUNG, Seoul National University — Recently, study on the insects' flapping flight became one of the challenging research subjects in the field of environmental engineering and 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 work, the unsteady flow physics around insect wings are conducted by utilizing numerical and computational simulation. The e-AIRS [6] (e-Science Aerospace Integrated Research System) gives a balanced service between computational and experimental aerodynamics, along with integrated research process of these two research activities. This paper presents the wing motions and their aerodynamics with a two dimensional approach to reduce environmental noise during the airflight. Also this paper shows an optimal phase angle, where the thrust is maximized at the position of minimized drag, which occurs when noise is minimized. Aside from the two-dimensional approach, stroke angles and phase angles of the airfoils are set as parameters, to determine which motion yields the best aerodynamic characteristics.

Sunday, November 24, 2013 7:00PM - 9:30PM –

Session F1 Reception Convention Center, Noresco Riverside Terrace and Exhibit Hall A -

7:00PM F1.00001 Reception –

Monday, November 25, 2013 8:00AM - 10:10AM –

Session G1 Geophysical: Oceanographic IV 323 - Matthew Paoletti, University of Texas at Austin

8:00AM G1.00001 Nonlinear generation of harmonics by an internal wave beam incident on a model oceanic pycnocline: numerical study[1] , PETER DIAMESSIS, Cornell University, SCOTT WUNSCH, IAN DELWICHE, Johns Hopkins University — The interaction of an internal wave beam with an idealized oceanic pycnocline is examined using 2-D fully nonlinear direct numerical simulation. These simulations vary the normalized pycnocline thickness and the ratio of peak pycnocline buoyancy frequency to that of the stratified lower layer. The angle of the beam with respect to the horizontal is held fixed at 45 degrees. Harmonics at the point of beam entry into the pycnocline increase in amplitude and number with the maximum gradient of the buoyancy frequency, suggesting refraction as an important factor in harmonic generation. For thin pynoclines, harmonics trapped within the pycnocline have maximum amplitude when their frequency and wavenumber match those of a natural pycnocline interfacial wave mode. For thicker pycnoclines, whose thickness is equal to the incident beam thickness, beam refraction results in harmonic generation at multiple locations in addition to pycnocline entry, giving rise to complex flow structure inside the pycnocline. These results are consistent with weakly nonlinear theory and with recent laboratory experiments.

[1]ONR grant N00014-08-1-0235 and NSF-CAREER award OCE-0845558.

8:13AM G1.00002 Nonlinear generation of harmonics by an internal wave beam incident on a model oceanic pycnocline: laboratory experiments , SCOTT WUNSCH, IAN DELWICHE, Johns Hopkins University, PETER DIAMESSIS, Cornell University — The interaction of an internal wave beam with an idealized oceanic pycnocline is examined using laboratory experiments. Laboratory results are compared to weakly nonlinear theory for a thin pycnocline, which succesfully predicts their qualitative features. The data show that harmonic modes with multiples of the incident frequency and wavenumber are generated near the point of pycnocline entry. For incidence angles exceeding 30 degrees, all harmonic modes are trapped within the pycnocline. Trapped harmonics appear to be strongest when their frequency and wavenumber match those of a natural pycnocline interfacial wave mode. For smaller incidence angles, the first harmonic radiates away from the pycnocline. These conclusions are supported by recent 2D numerical simlations. They also may be relevant to internal wave harmonic generation recently observed in the South China Sea and to the local generation of pycnocline internal solitary waves observed in the Bay of Biscay and elsewhere.

8:26AM G1.00003 Mixing by internal gravity waves that break at sloping topography , VAMSI CHALAMALLA, SUTANU SARKAR, University of California San Diego — Direct and large eddy simulations are performed to study the near-bottom mixing that occurs during the interaction of internal waves with a critical slope. The pathway from the input wave energy to the irreversible mixing of density field is explored. Diagnostics such as the turbulent kinetic energy budget and the density variance budget are discussed to explain the phasing of turbulence and associated mixing. Background and available potential energies are utilized to differentiate irreversible mixing from the reversible buoyancy flux. Mixing efficiency in all the simulated cases is found to be much higher than the frequently used value of 0.2 especially during large convective overturns. The ratio of Ozmidov and Thorpe length scales averaged over various sections of a wave cycle is investigated to assess inferences of turbulent dissipation rate from the Thorpe length scale.

8:39AM G1.00004 ABSTRACT WITHDRAWN –

8:52AM G1.00005 Resonant boundary currents fromtidal flow over topography need not generate intense internal waves[1] , HARRY SWINNEY, A. DETTNER, M.S. PAOLETTI, University of Texas at Austin — The relationship between boundary currents generated by tidal flow over topography and the radiated internal wave power is examined in two-dimensional numerical simulations for a uniformly stratified fluid. The radiated power and the kinetic energy density of the boundary currents are computed as a function of the internal wave slope and the criticality parameter (ratio of the maximum topographic slope to the internal wave slope). We consider cases where the hydrostatic approximation is valid as well as test theoretical predictions for models of the deep ocean where the beam slope diverges and thus the hydrostatic approximation fails. We confirm that resonant boundary currents with large kinetic energy densities form over critical topography. However, this resonance phenomenon does not extend to the power radiated by the internal waves that propagate away from the topography. The conclusion is that the kinetic energy density in the boundary currents cannot be used as a proxy to characterize the conversion of tidal energy to radiated internal wave power.

[1]Research supported by ONR MURI Grant N000141110701.

9:05AM G1.00006 Internal wave generation by tidal flow over topography in the deep ocean , MATTHEW S. PAOLETTI, MATTHEW C. DRAKE, HARRY L. SWINNEY, University of Texas at Austin — We present experimental and numerical studies of internal wave generation by tidal flow of an exponentially stratified fluid over bottom topography in a model of the deep ocean. King et al. (J. Geophys. Res. 117, C04008 (2012)) recently found many locations in the deep ocean where the stratification becomes so weak that the buoyancy frequency (proportional to the square root of the density gradient) becomes less than the tidal frequency; below such turning depths internal waves decay exponentially. Prior studies predict that topography beneath a turning depth would be unable to extract power from tidal motions and convert it to internal waves. However, we find that tidal motions over topography beneath a turning depth radiate internal waves, although the power is greatly diminished compared to cases of stronger stratification. We recover prior predictions of the radiated power by averaging the nonuniform stratification over an effective height. In the absence of a turning depth, the effective height is given by the actual topographic height, but for weak stratification where there is a turning depth, the effective height monotonically increases with turning depth height until it encompasses the entire fluid depth for very weak stratification.

9:18AM G1.00007 Determination of internal wave energy fluxes without pressure , FRANK M. LEE, M.S. PAOLETTI, HARRY L. SWINNEY, P.J. MORRISON, University of Texas at Austin — Internal waves are generated in the ocean by tidal flow over bottom topography, and they are of considerable interest because of their significant contribution to the energy budget of the ocean. However, the determination of the energy flux from real world data is difficult because knowledge of both the perturbation velocity and pressure fields is required. While the velocity perturbation field can be measured, e.g., by Particle Image Velocimetry, it is difficult to obtain simultaneous precision measurements of the pressure perturbation field. We propose a straightforward computational method to circumvent this difficulty for flows that are sufficiently 2-dimensional: the energy flux can be determined using only velocity data in the complete absence of pressure field knowledge. We demonstrate our method using laboratory and simulation data, and find that the conversion rate calculated from the flux field determined by the proposed method can be accurate to within 3 percent.

9:31AM G1.00008 Effect on Ex on internal waves created by tidal flow over near-critical topographic features , MASOUD JALALI B., NARSIMHA R. RAPAKA, SUTANU SARKAR, Mechanical and Aerospace Engineering, UC san Diego — Topographic bumps with small horizontal length under energetic surface tides with large velocity lead to internal tide generation in a regime with O(1) values of the excursion number, Ex, the ratio of fluid tidal advection to the topographic length scale. DNS is performed for a smoothed triangular ridge to study how internal gravity waves and turbulence change when Ex is varied from a low to O(1) values, keeping the Reynolds number constant. The near-field internal wave field looses it beam like character with increasing values of Ex. Analysis of the baroclinic energy shows significant reduction in the radiated wave flux higher Ex cases owing to a substantial rise in advection and baroclinic dissipation. There is small change in energy conversion consistent with the linear approximation. Turbulence changes qualitatively with increasing Ex. When $Ex \sim 0.1$, turbulence is intensified at the near-critical regions of the slope, and is also significant in the beams adjacent to the top of the ridge. However, at $Ex \sim 1$, turbulence is confined a narrow boundary region spanning the ridge and the adjacent flat bottom. The size of the turbulent overturns increases with increasing Ex until $Ex \sim 0.5$, followed by a substantial decrease.

9:44AM G1.00009 Energy Flux of Transmitted and Reflected Internal Waves , PRAJVALA KURTAKOTI, JAMES MUNROE, Memorial University of Newfoundland — As a preliminary step towards understanding how internal waves reflect off sloping topography such as the continental shelf, we performed a series of laboratory experiments to study the energy flux of propagating and reflecting internal waves in a continuously stratified salt water fluid. The internal waves are generated by a wave generator that is capable of producing monochromatic, vertically trapped waves. These internal waves propagate along the length of the tank (∼5m) and reflect. The structure and amplitude of the internal waves are measured using a technique called "synthetic schlieren" that also enables us to measure the energy in the wave. We examine how the vertical displacement amplitude and energy flux change as we change the frequency of the wave generator and the stratification of the fluid. Using Hilbert transform we have separated the waves traveling right to the waves traveling left enabling us to understand how much energy from the incoming waves is present in the reflected internal waves. The analysis of the energy flux of the internal waves during propagation and reflection using the Hilbert transform is helpful as it brings insight into phenomena that are difficult to observe during field studies.

9:57AM G1.00010 Subharmonic instability of locally confined internal wave beams[1] , HUSSAIN KARIMI, T.R. AKYLAS, MIT — Oceanic internal wave beams are central to tidal conversion—the transfer of tidal energy to internal waves by the interaction of the barotropic tide with sea-floor topography—a process believed to be important in deep-ocean mixing. There is evidence from recent experiments and numerical simulations that instability of internal wave beams is triggered by nonlinear interactions with fine-scale subharmonic disturbances. Motivated by these findings, we study analytically resonant triad interactions of a locally confined wave beam with small-amplitude, fine-scale, subharmonic disturbances for the purpose of discovering the conditions under which wave beams become unstable. A primary concern is to understand the dependence of the instability growth rates on beam amplitude and width. Furthermore, the effect of the Earth's rotation is investigated and possible resonant configurations are identified.

[1]Supported by NSF.

Monday, November 25, 2013 8:00AM - 10:10AM –

Session G2 Convection and Buoyancy-Driven Flows IV: Rayleigh-Benard Convection 324 - Richard Stevens, Johns Hopkins University

8:00AM G2.00001 Kinetic energy transport in Rayleigh-Bénard convection , KLAUS PETSCHEL, STEPHAN STELLMACH, Institut für Geophysik, Westfälische Wilhelms-Universität Münster, Germany, MICHAEL WILCZEK, Department of Mechanical Engineering, The Johns Hopkins University, Baltimore, USA, JOHANNES LÜLFF, Institut für Theoretische Physik, Westfälische Wilhelms-Universität Münster, Germany, ULRICH HANSEN, Institut für Geophysik, Westfälische Wilhelms-Universität Münster, Germany — Convective systems are often characterized by scaling laws for the heat transport. Several studies have indicated that these scaling laws are inextricably linked to the viscous dissipation rate and therefore to the kinetic energy balance. In the present study, direct numerical simulations of turbulent Rayleigh-Bénard convection are analyzed with respect to the horizontally averaged kinetic energy balance. Based on this budget equation, distinct regions where energy is produced, dissipated and transported by several flux processes are identified. These regions depend strongly on the Prandtl number, which gives new insights into the flow dynamics in the different Prandtl number regimes.

8:13AM G2.00002 Statistical description of Rayleigh-Bénard convection yields limit cycle behavior

, JOHANNES LUELFF, Institute for Theoretical Physics, WWU Muenster, MICHAEL WILCZEK, Johns Hopkins University Baltimore, RUDOLF FRIEDRICH, Institute for Theoretical Physics, WWU Muenster, RICHARD STEVENS, Johns Hopkins University Baltimore, DETLEF LOHSE, University of Twente — Rayleigh-Bénard convection describes the buoyancy-induced movement of a fluid enclosed between two horizontal plates, and serves as an idealized setup of phenomena occuring in nature and technical applications. The temperature fluctuations that occur in the fully turbulent case are of special interest, yet they can't be directly described from first principles due to the chaoticity of the system. Therefore we describe the statistics of temperature fluctuations by investigating the probability density function (PDF) of temperature, for the case of a cylindrical vessel and for periodic horizontal boundary conditions. Our ansatz is to derive exact evolution equations that describe the shape and deformation of the PDF; unclosed terms appearing in the form of conditional averages are estimated from direct numerical simulations of the two turbulent Rayleigh-Bénard systems. Following these steps, for both cases a limit cycle behavior appears in the phase space of the temperature PDF, highlighting the connection between the statistics and the dynamics of the system that our ansatz permits. The properties, interpretations and implications of this limit cycle are discussed; also, it is shown that the limit cycle can be connected to coherent structures formed by the convecting fluid.

8:26AM G2.00003 Resolving the fine-scale structure in turbulent Rayleigh-Benard convection

, JANET SCHEEL, Occidental College, Los Angeles, MOHAMMAD EMRAN, JOERG SCHUMACHER, Ilmenau University of Technology, Ilmenau, Germany — Results from high-resolution direct numerical simulations of turbulent Rayleigh-Benard convection in a cylindrical cell with an aspect ratio of one will be presented. We focus on the finest scales of convective turbulence, in particular the statistics of the kinetic energy and thermal dissipation rates in the bulk and the whole cell. These dissipation rates as well as the local dissipation scales are compared for different Rayleigh and Prandtl numbers. We also have investigated the convergence properties of our spectral element method and have found that both dissipation fields are very sensitive to insufficient resolution. We also demonstrate that global transport properties, such as the Nusselt number and the energy balances, are partly insensitive to insufficient resolution and yield consistent results even when the dissipation fields are under-resolved. Our present numerical framework is also compared with high-resolution simulations which use a finite difference method. For most of the compared quantities the agreement is found to be satisfactory.

8:39AM G2.00004 Mixed convection in a Rayleigh-Bénard cell with an imposed mean wind

, LAHCEN BOUHLALI, ANDREA SCAGLIARINI[1], HALLDÓR EINARSSON, ÁRMANN GYLFASON, School of Science and Engineering, Reykjavik University, Iceland, FEDERICO TOSCHI[2], Department of Mathematics and Computer Science, Eindhoven University of Technology, The Netherlands — Turbulent convection is present in a variety of natural occurring flows and engineering applications. In the most studied situation, the Rayleigh-Bénard (RB) setup, a fluid is confined between two differentially heated parallel plates under gravity. However, in many real-life situations, the picture can be complicated by flows interplaying/competing with the "natural" convection. In the atmosphere, for instance, thermal convection often coexists with currents due to pressure gradients. Buoyant and forced convection are also active in industrial flows (as in, e.g., heat exchangers). In this work we report a numerical study of a mixed convecting system. We consider a fully developed turbulent RB cell and at a given time we apply a constant pressure gradient, orthogonal to gravity. We will discuss the scaling properties of the heat flux with Rayleigh and friction Reynolds numbers as well as the statistics of small scale fluctuations of hydrodynamic fields. We will show that, depending on the relative ratio between buoyancy and pressure, the heat flux can be much depleted and the conductive profile for the temperature recovered. Such behaviour can be captured with simple phenomenological arguments. Comparisons with experimental results will be also presented.

[1]Eindhoven University of Technology, The Netherlands
[2]Department of Applied Physics, Eindhoven University of Technology, The Netherlands

8:52AM G2.00005 Statistical classification of flow morphology in rapidly rotating Rayleigh-Bénard convection: A numerical and experimental synthesis[1]

, DAVID NIEVES, ANTONIO RUBIO, KEITH JULIEN, University of Colorado at Boulder — We use experimentally accessible statistical measures to distinguish between flow morphologies in rapidly rotating Rayleigh-Bénard convection (RRBC). Transitions between different flow regimes are identified for the fixed non-dimensional Prandtl number $\sigma = 7$ in terms of the reduced Rayleigh number $\widetilde{Ra} = RaE^{4/3}$, where E is the non-dimensional Ekman number. Using cross-correlations of synthetic thermistor time signals we find that the flow transitions from the cellular regime to the convective Taylor column (CTC) regime at $\widetilde{Ra} \approx 20$, and from the CTC regime to the plume regime at $\widetilde{Ra} \approx 57$. Additionally, the horizontal flow structure is elucidated via spatial cross-correlations of vertically separated thermal fluctuations. Length, time, and velocity scales are produced for coherent columnar structures via spatial and temporal cross-correlations. Length, time and velocity scale data is seen to fit power-laws of the form $\alpha(\widetilde{Ra} - \widetilde{Ra_c})^{\beta}$, where $\widetilde{Ra_c}$ is the critical Rayleigh number for the onset of stationary convection. Through direct numerical simulation of non-hydrostatic quasi-geostrophic equations, a detailed examination of the flow morphology in RRBC is carried out.

[1]NSF FRG No. DMS-0855010

9:05AM G2.00006 Active transport in chaotic Rayleigh-Bénard convection

, CHRISTOPHER MEHRVARZI, MARK PAUL, Virginia Polytechnic Institute and State University — The active transport of a scalar species is studied numerically in a spatiotemporally chaotic flow field of Rayleigh-Bénard convection. There has been significant progress both theoretically and experimentally in understanding characteristics of active transport in steady periodic-flows such as a ring of vortices and other two-dimensional flows. In this work we are interested in the reaction-advection-diffusion of a scalar species in a three-dimensional chaotic flow field that is accessible to the laboratory. We study the transport using a highly efficient and parallel spectral element approach to simultaneously evolve the Boussinesq and reaction-advection-diffusion equations in large aspect-ratio cylindrical domains with experimentally relevant boundary conditions. We choose the system parameters to yield advection, reaction, and diffusion time scales that are comparable and investigate their interactions. We explore the effect of the chaotic convection patterns on the transport characteristics and quantify the reaction front speed and front geometry for a range of parameters.

9:18AM G2.00007 Size-Dependent Rayleigh–Bénard Problem

, AREZOO HAJESFANDIARI, ALI HADJESFANDIARI, GARY DARGUSH, University at Buffalo, State University of New York — Problems of thermoviscous flows are of prime importance for many physical processes. Here the classical Boussinesq equations are modified by including couple stresses, which account for size-dependency. This size-dependency is specified by a material length scale l, which becomes increasingly important as the characteristic geometric dimension of the problem decreases. The modified two-dimensional linear momentum equations become

$$\rho\left(\frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y}\right) = -\frac{\partial p}{\partial x} + \mu\nabla^2 u - \mu l^2\nabla^2\nabla^2 u$$

$$\rho\left(\frac{\partial v}{\partial t} + u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y}\right) = -\frac{\partial p}{\partial y} + \mu\nabla^2 v - \mu l^2\nabla^2\nabla^2 v - \rho\alpha\left(T - T_0\right)$$

The stability of natural convection for the Rayleigh–Bénard problem is studied numerically and we consider the onset of convective instability and multiple stable steady states arising within specific ranges of Rayleigh and Prandtl numbers and l.

9:31AM G2.00008 Boiling Rayleigh-Benard flow[1] , DANIELA NAREZO, U. Twente, Netherlands / UCSB, Santa Barbara, USA, YANBO XIE, U. Twente, Netherlands, GUENTER AHLERS, UCSB, Santa Barbara, USA, CHAO SUN, DETLEF LOHSE, U. Twente, Netherlands — We report on heat transport due to boiling of Novec7000 (1-methoxyheptafluoropropane) at the bottom plate of a turbulent Rayleigh-Benard sample which was filled with liquid (except for small vapor bubbles when boiling took place). The top surface of the bottom plate was a silicone wafer covered by a triangular lattice of 30 μm diameter and 100 μm deep cavities (the lattice spacing was 100 μm). The plate diameter and sample height both were 10 cm, but only a central bottom-plate area of 2.5 cm diameter was heated. When the cavities were activated (deactivated) by assuring that they were filled by vapor (liquid), then they nucleated (did not nucleate) bubble formation for bottom-plate temperatures T_b larger than the boiling point T_{BP}. Results of the heat transport and of the mid height temperature at the side wall of the sample as a function of T_b with a fixed applied temperature difference $\Delta T = T_b - T_t = 20$K ($T_t$ is the top plate temperature) will be reported. When $T_b > T_{BP}$, the effective conductivity of the 2-phase flow was enhanced relative to the supersaturated 1-phase system by up to 40 percent. The sidewall temperature T_s was reduced in the presence of bubbles by up to 3 percent relative to the 1-phase case.

[1]Work supported by an ERC-Advanced Grant and by NSF grant DMR11-58514.

9:44AM G2.00009 Periodic mode competition in Rayleigh-Benard convections with a horizontal magnetic field , YUJI TASAKA, KAZUTO IGAKI, Hokkaido University, TAKATOSHI YANAGISAWA, IFREE, JAMSTEC, SVEN ECKERT, HZDR — Recent experimental studies (Yanagisawa, *et al.*, 2011) indicated that appearance of random flow reversals in Rayleigh-Benard convection with a horizontal magnetic field. Time intervals of the flow reversals obey Poisson process and this indicates that this event is memoryless. Bi-stable nature of this system under a condition of non-dimensional parameters, Rayleigh number Ra and Chandrasekhar number Q, may induce this event with influences of external random noise. This even appears around $Ra = 10Q$ in a range, $5 \times 10^2 < Q < 10^3$, where the upper limit is determined by the maximum intensity of magnetic field. The present study investigates extrapolability of this relation in a higher range of Q up to $Q = 10^4$. The test fluid container has dimension of square of 200 mm in the horizontal plane and 40 mm in height. The container was filled with GaInSn and ultrasonic velocity profiling achieved quantitative flow pattern visualization. The visualization confirmed the extrapolability of the relation on flow reversals, but the observed flow reversals were not random but periodic. Proper orthogonal decomposition on the space-time velocity map elucidated periodic competitions between two convection modes with different wavenumbers in the periodic flow reversals.

9:57AM G2.00010 Torsional oscillation of the large-scale circulation in turbulent Rayleigh-Bénard convection at large Rayleigh numbers[1] , DENNIS P.M. VAN GILS, XIAOZHOU HE, MPI-DS, Göttingen, Germany, GUENTER AHLERS, UCSB, Santa Barbara, USA, EBERHARD BODENSCHATZ, MPI-DS, Göttingen, Germany — We present temperature measurements in turbulent Rayleigh–Bénard convection (RBC) over the Rayleigh number range $3.0 \times 10^{13} \leq Ra \leq 1.3 \times 10^{14}$ and at constant Prandtl number $\mathrm{Pr} \approx 0.8$. The RBC sample, known as the High-Pressure Convection Facility (HPCF) of Göttingen [1], is an upright cylinder of aspect ratio $\Gamma = 1.00$. Using three horizontal rows of thermistors at different heights in the sample, we determined the orientation angle of the large-scale circulation (LSC) plane, similar to [2]. Results identify a well established single-roll LSC with a periodic "torsional" mode with a frequency f_C. The values of f_C are consistent with the frequencies f_L obtained from power spectra $P(f)$ of temperature time series taken at mid-height of the sample. The non-dimensionalized frequencies $\tilde{f}_C$ are well described by a power law: $\tilde{f}_C \propto Ra^{\zeta_f}$ with $\zeta_f = 0.427 \pm 0.001$.

[1] He, Funfschilling, Bodenschatz and Ahlers, New J. Phys. **14**, 063030 (2012).
[2] Weiss and Ahlers, J. Fluid Mech. **688**, 461 (2011).

[1]Supported by the Max Planck Society, the Volkswagen Stiftung, the DFD Sonderforschungsbereich SFB963, and NSF grant DMR11-58514.

Monday, November 25, 2013 8:00AM - 9:57AM –

Session G3 Multiphase Flows IV 325 - Sean Garrick, University of Minnesota

8:00AM G3.00001 Simulation of sprays using a Lagrangian filtered density function approach , WANJIAO LIU, SEAN GARRICK, University of Minnesota — Sprays and atomization have wide applications in industry, including combustion/engines, pharmaceutics and agricultural spraying. Due to the complexity of the underlying processes, much of the underlying phenomena are not fully understood. Numerical simulation may provide ways to investigate atomization and spray dynamics. Large eddy simulation (LES) is a practical approach to flow simulation as it resolves only the large-scale structures while modeling the sub-grid scale (SGS) effects. We combine a filtered density function (FDF) based approach with a Lagrangian volume-of-fluid method to perform LES. This resulting methodology is advantageous in that it has no diffusive or dissipative numerical errors, and the highly non-linear surface tension force appears in closed form thus the modeling of the SGS surface tension is not needed when simulating turbulent, multiphase flows. We present the methodology and some results for the simulation of multiphase jets.

8:13AM G3.00002 Eulerian CFD modeling and X-ray validation of non-evaporating diesel spray , QINGLUAN XUE, SIBENDU SOM, Argonne National Laboratory, SHAOPING QUAN, ERIC POMRANING, P.K. SENECAL, Convergent Science Inc. — This work implemented an Eulerian single-phase approach by Vallet et al. [1] into CFD software (Convergent) for diesel spray simulations. This Eulerian approach considers liquid and gas phase as a complex mixture of a single flow with a highly variable density to describe the near nozzle dense sprays. The mean density is obtained form the Favre-averaged liquid mass fraction. Liquid mass fraction is transported with a model for the turbulent liquid diffusion flux into the gas. A mean gradient-based model is employed for the diffusion flux in this study. A non-evaporating diesel spray was measured using x-ray radiography at Argonne National Laboratory. The quantitative and time-resolved data of liquid penetration and mass distribution in the dense spray region are used to validate this approach. The different turbulence models are also used for the simulations. The comparison between the simulated results and experimental data and the turbulence model effect are discussed.

[1] Vallet et al., Atomization and Sprays, vol. 11, pp. 619-642, 2001.

8:26AM G3.00003 LES/FMDF of High Speed Spray Combustion , ABOLFAZL IRANNEJAD, FARHAD JABERI, Michigan State University — High speed evaporating and combusting sprays are computed with the hybrid two-phase large eddy simulation (LES)/filtered mass density function (FMDF) methodology. In this methodology, the resolved fluid velocity is obtained by solving the filtered form of the compressible Navier-Stokes equations with high-order finite difference schemes. The scalar (temperature and species mass fractions) field is obtained by solving the FMDF transport equation with a Lagrangian stochastic method. The spray is simulated with the Lagrangian droplets together with stochastic breakup and finite rate heat and mass transfer models. The liquid volume fraction is included in the LES/FMDF for denser spray regions. Simulations of high speed evaporating sprays with and without combustion for a range of gas and spray conditions indicate that the two-phase LES/FMDF results are consistent and compare well with the experimental results for global spray variables such as the spray penetration and flame lift-off lengths. The gas velocity and turbulence generated by the spray are found to be very significant in all simulated cases. A broad spectrum of droplet sizes is also found to be generated by the complex and coupled effects of the gas flow turbulence, droplet breakup, evaporation and combustion.

8:39AM G3.00004 Analysis of an Electrostatic Spray Injector , MATTHEW RYAN, JONATHAN TENNIS, United States Military Academy, CHOL-BUM KWEON, United States Army Research Laboratory, MICHAEL BENSON, BRET VAN POPPEL, United States Military Academy — The objective of the current research is to assess the effects of electrostatic injector designs and charge energy on the spray break-up process. Electrostatic injectors have a potential to improve the liquid fuel spray atomization at low fuel pressures. The application areas include carbureted and port fuel injections in small engines to improve fuel-air mixing. In this study, two different electrostatic injector designs are tested in an ambient chamber with four optical windows. Shadowgraphy and Mie scattering techniques are used to measure the major spray parameters such as spray patterns, spray angles, and liquid fuel penetration length. Shadowgraphy with a micro zoom lens is used to measure droplet distributions and droplet pressures. An analysis of the results is presented to inform electrostatic injector design.

8:52AM G3.00005 ABSTRACT WITHDRAWN –

9:05AM G3.00006 Direct numerical simulation of leaky dielectrics with application to electrohydrodynamic atomization , MARK OWKES, OLIVIER DESJARDINS, Cornell University — Electrohydrodynamics (EHD) have the potential to greatly enhance liquid break-up, as demonstrated in numerical simulations by Van Poppel et al. (JCP (229) 2010). In liquid-gas EHD flows, the ratio of charge mobility to charge convection timescales can be used to determine whether the charge can be assumed to exist in the bulk of the liquid or at the surface only. However, for EHD-aided fuel injection applications, these timescales are of similar magnitude and charge mobility within the fluid might need to be accounted for explicitly. In this work, a computational approach for simulating two-phase EHD flows including the charge transport equation is presented. Under certain assumptions compatible with a leaky dielectric model, charge transport simplifies to a scalar transport equation that is only defined in the liquid phase, where electric charges are present. To ensure consistency with interfacial transport, the charge equation is solved using a semi-Lagrangian geometric transport approach, similar to the method proposed by Le Chenadec and Pitsch (JCP (233) 2013). This methodology is then applied to EHD atomization of a liquid kerosene jet, and compared to results produced under the assumption of a bulk volumetric charge.

9:18AM G3.00007 ABSTRACT WITHDRAWN –

9:31AM G3.00008 Large Scale Behavior and Droplet Size Distributions in Crude Oil Jets and Plumes[1] , JOSEPH KATZ, DAVID MURPHY, DAVID MORRA, Johns Hopkins University — The 2010 *Deepwater Horizon* blowout introduced several million barrels of crude oil into the Gulf of Mexico. Injected initially as a turbulent jet containing crude oil and gas, the spill caused formation of a subsurface plume stretching for tens of miles. The behavior of such buoyant multiphase plumes depends on several factors, such as the oil droplet and bubble size distributions, current speed, and ambient stratification. While large droplets quickly rise to the surface, fine ones together with entrained seawater form intrusion layers. Many elements of the physics of droplet formation by an immiscible turbulent jet and their resulting size distribution have not been elucidated, but are known to be significantly influenced by the addition of dispersants, which vary the Weber Number by orders of magnitude. We present experimental high speed visualizations of turbulent jets of sweet petroleum crude oil (MC 252) premixed with Corexit 9500A dispersant at various dispersant to oil ratios. Observations were conducted in a 0.9 m x 0.9 m x 2.5 m towing tank, where large-scale behavior of the jet, both stationary and towed at various speeds to simulate cross-flow, have been recorded at high speed. Preliminary data on oil droplet size and spatial distributions were also measured using a videoscope and pulsed light sheet.

[1]Sponsored by Gulf of Mexico Research Initiative (GoMRI)

9:44AM G3.00009 Experimental and computational investigation of underwater buoyant oil jets , LEANDRE BERARD, MEHDI RAESSI, MICHAEL BAUER, PETER FRIEDMAN, STEPHEN CODYER, University of Massachusetts Dartmouth, UNIVERSITY OF MASSACHUSETTS DARTMOUTH TEAM — We present experimental and numerical results on the breakup of underwater positively buoyant oil jets at a wide range of Reynolds, Weber and Richardson numbers and several viscosity ratios. Three main jet breakup regimes are observed: atomization, skirt-type and pinch-off. A threshold Weber number for the atomization regime is found to be around 100, varying slightly with the jet Eötvös number. The Ohnesorge-Reynolds correlation proposed by Masutani and Adams as the boundary for the atomization regime is shown to be applicable to our broader data set as well. Results suggest that the breakup of a positive buoyancy-driven jet occurs only when the jet is accelerated to a point where the local Richardson number becomes less than 0.4, in which case the local Weber number is above 10. The numerical results reveal the mechanisms leading to formation of small droplets around the perimeter of energetic jets and umbrella-shaped jet separations at less energetic cases. The time-averaged lateral expansion of the simulated jets, representing four different conditions, are presented as a function of the height along the jet.

Monday, November 25, 2013 8:00AM - 9:44AM –

Session G4 Separated Flows I: Diffusers and Massively Separated Flows 326 -

8:00AM G4.00001 Asymmetric Separation and Perturbation Sensitivity in an Annular Diffuser , JESSE COFFMAN, SCOTT MORRIS, ALEKSANDER JEMCOV, JOSHUA CAMERON, University of Notre Dame — When an annular diffuser stalls, the separation can take many forms. Experiments show that one type of separation appears to be asymmetric and periodic. This asymmetry appears to be influenced by upstream and downstream components and inlet flow conditions. By understanding the changes effected at the exit of the diffuser by the inlet perturbations, the diffuser performance can be more accurately predicted within a system. This work aims to understand the influence of velocity perturbations at the inlet of the diffuser on the overall duct performance. This is done by application of the Euler equations and a RANS simulation for various circumferential wavenumbers.

8:13AM G4.00002 Multiple local recirculations to reduce flow separation and increase efficiency of diffusers , ALESSANDRO MARIOTTI, GUIDO BURESTI, MARIA VITTORIA SALVETTI, DICI, University of Pisa — Boundary layer separation causes a decrease of performance in diffusers. We analyze a passive method for flow control in diffusers, based on the introduction of single and multiple contoured cavities in the walls. The proposed control strategy is investigated through numerical simulations of the flow in plane diffusers having an area ratio of 2 and different divergence angles. The location and geometry of the cavities are numerically optimized to maximize the pressure recovery in the diffuser. In all cases, the introduction of the optimal cavities leads to a strong increase in the pressure recovery and, when present, to a significant reduction of the main flow separation zone. The flow separates at the cavity upstream edge and rapidly reattaches, forming a small closed recirculation region within and immediately downstream of the cavities. These recirculation zones lead to both a favorable local modification of the virtual shape of the diffuser and to a reduction of the dissipation in the near-wall region due to the relaxation of the no-slip condition. A generic optimization of the diffuser shape is also carried out; if the number of degrees of freedom is large enough, the presence of small local recirculations is again found in the optimized configuration.

8:26AM G4.00003 Unsteady Structure of Three-Dimensional Stall Cells[1] , KEVIN DISOTELL[2], JAMES GREGORY[3], The Ohio State University — A description of three-dimensional flow separation encountered on wings operating just above their maximum lift condition remains a critical link toward increased payload capability for air vehicles. In particular, the development of spanwise-periodic separation zones or "stall cells" observed in the surface streamline pattern appear to result from the amplification of a spanwise instability in the separated shear layer for certain stall types. Time-averaged point measurements and steady simulations have largely been used to characterize the geometry of stall cells, although unsteady motion due to possible shear layer flapping has been reported by Yon and Katz (1998). Details of the flow topology remain unclear especially under turbulent flow conditions. The development of stall cells on a rectangular NACA 0015 airfoil of aspect ratio 2.5 at chord Reynolds numbers above 750,000 is investigated in the current work using planar, time-resolved particle image velocimetry measurements. Flow visualization with miniature surface tufts and high-speed imaging revealed intermittent cellular patterns at incipient stall conditions which are explored further. Understanding the behavior of cellular separation can help inform flow control strategies aimed at mitigating stall.

[1]K. Disotell is supported by a NSF Graduate Research Fellowship.
[2]Ph.D. Candidate, Dept. of Mechanical and Aerospace Engineering
[3]Assistant Professor, Dept. of Mechanical and Aerospace Engineering

8:39AM G4.00004 Turbulence characteristics of separated boundary layer flow under unsteady pressure gradients using direct numerical simulation[1] , JUNSHIN PARK, Department of Mechanical Engineering, Pohang University of Science and Technology, WILLIAM BROMBY, DONGHYUN YOU, Department of Mechanical Engineering, Carnegie Mellon University — To understand turbulence characteristics of separated boundary layer flow under unsteady pressure gradients, a direct numerical simulation study is performed. Steady and unsteady blowing-suction velocity distributions are imposed along the upper boundary of the computational domain to introduce steady and unsteady adverse pressure gradients leading to steady and unsteady separated turbulent boundary layers, respectively. Time averaged and phase averaged turbulence statistics such as velocity, vorticity, kinetic energy budgets, Reynolds stress budgets, wall pressure fluctuations and skin friction distributions are examined in detail with aims of gaining understanding of flow physics for unsteady separated turbulent boundary layer and the sources of incapability of the conventional Reynolds-averaged Navier-Stokes models in predicting unsteady separation.

[1]Supported by the Army Research Office Grant W911NF1010348 and the National Research Foundation of Korea Grant NRF-2012R1A1A2003699

8:52AM G4.00005 ABSTRACT WITHDRAWN –

9:05AM G4.00006 Spanwise correlation lengths of unsteady surface pressure behind a backward facing step , MICHAEL BILKA, MARK PALUTA, SCOTT MORRIS, University of Notre Dame — The flow over a backward facing step is a canonical test case used for the study of separated flow problems. Study of this configuration has led to deeper understanding of the behavior and structure of separated flows. For many practical applications, it is not only necessary to understand the flow behavior but also the generated unsteady surface pressure. This information can provide a basis for understanding the unsteady loading experienced by a body, which can generate unwanted flow-structure interactions, such as vibration and sound radiation. The present work investigates the unsteady surface pressure generated by the separated flow behind a backwards facing step. Using an extensive array of surface pressure sensors, measurement is made not only of the unsteady surface pressure but also of the spanwise correlation of the surface pressure fluctuations at various locations of separation and reattachment. The spanwise correlation has received limited treatment in the literature for separated flows and is important for the modeling and prediction of fluid-structure interactions.

9:18AM G4.00007 Flow over a backward-facing step: Mean separation bubble and evolution of coherent structures , PANKAJ NADGE, RAGHURAMAN GOVARDHAN, Indian Institute of Science — We present PIV measurements downstream of a backward-facing step at large step based Reynolds numbers. The structure of the mean separation bubble is mapped in detail, and the effect of Reynolds number and expansion ratio (ER) on it is studied; the ER being the primary geometrical parameter for this configuration. These measurements show that there exists a mean separation bubble structure that is nearly independent of ER at large Re. Further, these measurements permit evaluation of the forces acting on the mean separation bubble in the streamwise direction due to the Reynolds stresses. Towards understanding the coherent structures in the flow downstream of the step, time-resolved PIV measurements have been performed in a plane parallel to the lower wall. These show the presence of counter-rotating vortical structures, which may be thought of as signatures of three-dimensional hairpin-like structures. These counter-rotating pairs are observed to evolve as they convect downstream. Conditional averaging of these counter-rotating structures show that their length-scale increases with streamwise distance. Details about these structures and their evolution will be presented at the conference.

9:31AM G4.00008 Data Reduction Methods to Identify Characteristic Scales in Transient, Inhomogeneous Flows , JOHN DANTONIO, JOSHUA CAMERON, SCOTT MORRIS, University of Notre Dame — Identification of meaningful spatial and temporal scales in transient, inhomogeneous flows is challenging as many data reduction methods favor spatial over temporal resolution (or vice versa). Spectra of state variables often contain significant broadband content and may be non-stationary. Measurements may be over an inhomogeneous grid or have unknown convergence properties. Correlation and wavelet based methods are presented to identify both spatial and temporal scales from pseudo-random data. These methods are relevant to many separated industrial flows; an application to axial compressor rotating stall will be presented.

Monday, November 25, 2013 8:00AM - 10:10AM –

Session G5 CFD IV 327 - S. Levent Yilmaz, MathWorks

8:00AM G5.00001 2D Unstructured Finite Volume Lattice Boltzmann Model for Flow with Complex Geometric Boundaries , LEITAO CHEN, LAURA SCHAEFER, University of Pittsburgh — Many of the numerical issues of LBM (lattice Boltzmann method) are not yet fully solved. One of the issues is its inability of handling complex geometric boundaries. Some published work, which is based on collision-streaming discretization of the LBE and corresponding lattice-like mesh, introduced successful treatments for curved boundaries. However, those schemes are not applicable to the boundaries with large curvature like porous media since the lattice-like mesh is not able to recognize it. In order to solve this issue, a 2D FVM (finite volume method)-based numerical framework is proposed, which completely uncouples the lattice structure and the spatial discretization and therefore brings the freedom of using any type of lattice structure while keeping the basic framework unchanged. The model is solved on an unstructured triangular mesh and triangular control volume. Boundary schemes of isothermal and thermal flow for the new numerical framework are also studied. Finally, a variety of isothermal and thermal flow problems are simulated and compared with other work. The proposed model can simulate the flow with a complex geometry to the desired accuracy in addition to complementing the simple geometry of the existing LB model.

8:13AM G5.00002 A Second-Order Finite-Difference Scheme for the Lattice Boltzmann Method , PARTHIB RAO, LAURA SCHAEFER, University of Pittsburgh — The lattice Boltzmann method (LBM) is being increasingly used as an alternative solver for the isothermal Navier-Stokes equation, as well as for other complex flows. However, due to an innate coupling between the velocity and the configuration space, LBM is restricted to uniform grids. This is a serious impediment for simulating flows with large gradients, flow around objects, etc. The discrete Boltzmann-BGK equation, which forms the basis of LBM, can be viewed as a set of hyperbolic equations with constant coefficients and a source term. We, therefore, use the finite difference method to discretize the Boltzmann-BGK equation (FDLBM). In FDLBM, the velocity-lattice is uncoupled from the spatial lattice allowing us to choose discrete velocities and space-time steps independently. The currently available FDLBM models have either narrow a stability range, or have large computational costs. To overcome these constraints, we employ the Lax-Wendroff scheme for the advection part, and central-difference for the spatial gradients, resulting in a scheme that is both explicit and second-order in both space and time. The proposed scheme is validated for an isothermal incompressible lid-driven cavity flow. The results indicate improved stability (in terms of CFL conditions) compared to the current explicit FDLB models, due to the addition of the second-order temporal terms. The maximum Reynolds number that can be simulated stably is also much higher. The relationship between the discrete time-step and the relaxation parameter, and extension of the FDLBM to a non-uniform mesh are also discussed.

8:26AM G5.00003 A Block-Structured Adaptive Mesh Refinement Technique with a Finite-Difference-Based Lattice Boltzmann Method , ABBAS FAKHARI, TAEHUN LEE, City College of the City University of New York — A novel adaptive mesh refinement (AMR) algorithm for the numerical solution of fluid flow problems is presented in this study. The proposed AMR algorithm can be used to solve partial differential equations including, but not limited to, the Navier-Stokes equations using an AMR technique. Here, the lattice Boltzmann method (LBM) is employed as a substitute of the nearly incompressible Navier-Stokes equations. Besides its simplicity, the proposed AMR algorithm is straightforward and yet efficient. The idea is to remove the need for a tree-type data structure by using the pointer attributes in a unique way, along with an appropriate adjustment of the child block's IDs, to determine the neighbors of a certain block. Thanks to the unique way of invoking pointers, there is no need to construct a quad-tree (in 2D) or oct-tree (in 3D) data structure for maintaining the connectivity data between different blocks. As a result, the memory and time required for tree traversal are completely eliminated, leaving us with a clean and efficient algorithm that is easier to implement and use on parallel machines. Several benchmark studies are carried out to assess the accuracy and efficiency of the proposed AMR-LBM, including lid-driven cavity flow, vortex shedding past a square cylinder, and Kelvin-Helmholtz instability for single-phase and multiphase fluids.

8:39AM G5.00004 Numerical investigations on the vortex-induced vibration of moving rigid body by using the Lattice Boltzmann Method[1] , XIAOHAI JIANG[2], TAEHUN LEE, YIANNIS ANDREOPOULOS, ZHEXUAN WANG, Department of Mechanical Engineering, City College of City University of New York, New York, 10031, USA — Vortex-induced vibrations (VIV) phenomena related to self-excited energy harvesters consisting of circular or square cylinders have been investigated numerically by using the BGK or MRT Lattice Boltzmann Method. In the present work such a harvester is placed inside a channel flow and is allowed to oscillate without a structural restoring force in a direction normal to the flow. Currently the half-way bounce-back boundary scheme and interpolations are being used to model the moving boundary. The numerical results were compared to the ones by classical CFD methods and experiments. A good agreement was obtained. The vortex dynamics and the development of the flow patterns for different flow parameters such as Reynolds number, blockage and aspect ratios will be presented. Particular emphasis is given to the dynamics of vortex pairing observed in several of the simulations. The present approach will be extended to simulate the flexible beam with the Immersed Boundary Method.

[1]Sponsored by the National Science Foundation (CBET #1033117) and a fellowship support from China Scholarship Council.
[2]Key Laboratory of Transient Physics, Nanjing University of Science and Technology, Nanjing, 210094, China

8:52AM G5.00005 Isothermal Multiphase Flow using a Multi-domain Lattice Boltzmann Method , CHRISTOPHER J. FORSTER, MARC K. SMITH, Georgia Institute of Technology — In an effort to increase the useful property range of Lattice Boltzmann multiphase flow simulations, a multiple fluid domain approach has been developed. Specifically, the purpose of this approach is to allow higher density and viscosity ratios across fluid interfaces with minimal spurious currents and instability. The multiple domains are coupled through interpolated boundary conditions. A level-set method on a collocated grid is used to track the interface location, which provides the necessary information for implementing moving, interpolated boundary conditions on each of the domains. The multi-domain Lattice Boltzmann method coupled with a level-set method allows for a sharp interface to apply the interfacial conditions and surface tension forces, while implicitly handling topological changes. To demonstrate the capabilities of this method, a test case of buoyancy driven bubble train flow will be presented with several increasing density and viscosity ratios.

9:05AM G5.00006 A Highly-Parallelized Perfectly Stirred Reactor (PSR) Model Using GPU Acceleration , SUDIP ADHIKARI, ABHILASH J. CHANDY, Department of Mechanical Engineering, University of Akron, Akron — Perfectly stirred reactors (PSR), which are idealized systems, where species undergoing chemical reactions have high rate of mixing, have been found to be very useful in testing and developing chemical reaction mechanisms for combustion research. The PSR model requires solving systems of nonlinear algebraic equations governing the chemical reactions, which typically are of the order of hundreds for realistic engineering systems and also involve multiple time scales ranging over a few orders of magnitude. As a result, the equations are stiff and the solution is highly compute-intensive. In spite of dramatic improvements in central processing units (CPUs) made during the past several decades, PSR solutions, while they remain feasible are computationally very expensive. An alternative approach is the application of accelerator technologies, such as graphics processing units (GPUs) that can improve the performance of such algorithms. A highly parallelized GPU implementation is presented for the PSR model, using a robust and efficient non-linear solver. Parallel performance metrics are presented to demonstrate the capability of GPUs to accelerate chemical kinetics calculations.

9:18AM G5.00007 Asynchronous schemes for CFD at extreme scales , ADITYA KONDURI, DIEGO DONZIS, Texas A&M University — Recent advances in computing hardware and software have made simulations an indispensable research tool in understanding fluid flow phenomena in complex conditions at great detail. Due to the nonlinear nature of the governing NS equations, simulations of high Re turbulent flows are computationally very expensive and demand for extreme levels of parallelism. Current large simulations are being done on hundreds of thousands of processing elements (PEs). Benchmarks from these simulations show that communication between PEs take a substantial amount of time, overwhelming the compute time, resulting in substantial waste in compute cycles as PEs remain idle. We investigate a novel approach based on widely used finite-difference schemes in which computations are carried out asynchronously, i.e. synchronization of data among PEs is not enforced and computations proceed regardless of the status of messages. This drastically reduces PE idle time and results in much larger computation rates. We show that while these schemes remain stable, their accuracy is significantly affected. We present new schemes that maintain accuracy under asynchronous conditions and provide a viable path towards exascale computing. Performance of these schemes will be shown for simple models like Burgers' equation.

9:31AM G5.00008 A Parallel Hexahedral Unstructured Adaptive Mesh Refinement Library , CARLOS BALLESTEROS, MARCUS HERRMANN, Arizona State University — Adaptive mesh refinement (AMR) libraries can simplify the generation of meshes surrounding complex or moving boundaries, as well as focus computational resources only in the areas of the solution domain that are of interest through the use of recursive cell refinement. By applying AMR within an unstructured hexahedral mesh framework, the resulting mesh retains the favorable numerical properties of hexahedral elements, while possessing characteristics advantageous for usage in high-performance computing. These properties include straightforward refinement and coarsening operations; as well as explicit connectivity between solution cells, which make neighbor-cell lookups, domain decomposition and load balancing simple, especially when compared with tree AMR approaches. The parallel scalability of a unstructured hexahedral AMR library, FARCOM, will be presented, with its ability to generate meshes illustrated with several test cases. Additionally, extensions to convection-diffusion, incompressible flow, and immersed-boundary problems will be discussed.

9:44AM G5.00009 Parallel Cartesian grid refinement for 3D complex flow simulations[1] , DIONYSIOS ANGELIDIS, FOTIS SOTIROPOULOS, St. Anthony Falls Laboratory, Department of Civil Engineering, 2 Third Avenue SE, Minneapolis, MN 55414, USA — A second order accurate method for discretizing the Navier-Stokes equations on 3D unstructured Cartesian grids is presented. Although the grid generator is based on the oct-tree hierarchical method, fully unstructured data-structure is adopted enabling robust calculations for incompressible flows, avoiding both the need of synchronization of the solution between different levels of refinement and usage of prolongation/restriction operators. The current solver implements a hybrid staggered/non-staggered grid layout, employing the implicit fractional step method to satisfy the continuity equation. The pressure-Poisson equation is discretized by using a novel second order fully implicit scheme for unstructured Cartesian grids and solved using an efficient Krylov subspace solver. The momentum equation is also discretized with second order accuracy and the high performance Newton-Krylov method is used for integrating them in time. Neumann and Dirichlet conditions are used to validate the Poisson solver against analytical functions and grid refinement results to a significant reduction of the solution error. The effectiveness of the fractional step method results in the stability of the overall algorithm and enables the performance of accurate multi-resolution real life simulations.

[1]This material is based upon work supported by the Department of Energy under Award Number DE-EE0005482.

9:57AM G5.00010 Domain decomposition for coupled Stokes and Darcy flows with floating Stokes domains , CHANGQING WANG, IVAN YOTOV, University of Pittsburgh — A non-overlapping domain decomposition method is presented to solve a coupled Stokes-Darcy flow problem in parallel by partitioning the computational domain into multiple subdomains. Specifically, in the case where floating Stokes subdomain occurs, an approach based on the FETI methods is introduced and tested.

Monday, November 25, 2013 8:00AM - 10:10AM –

Session G6 Microfluids: Particles II - Electrokinetically Induced Flow 328 - German Drazer, Rutgers University

8:00AM G6.00001 Direct numerical simulations (DNS) of particles in spatially varying electric fields , E. AMAH, NJIT, M. JANJUA, American University in Dubai, I.S. FISCHER, P. SINGH, NJIT — We have developed a direct numerical simulation (DNS) scheme to simulate the motion of dielectric particles suspended in a dielectric liquid in nonuniform electric fields. The motion of particles is tracked using a distributed Lagrange multiplier method (DLM) and the electric forces acting on the particles are calculated by an efficient scheme in which the Maxwell stress tensor (MST) is integrated over the surfaces of the particles to obtain the force. The code is validated by performing a convergence study and by comparing the particle trajectories in a dielectrophoretic cage with those given by the point-dipole method. We also show that the trajectories of the two or more interacting particles given by the MST method can be different from those obtained using the point-dipole method since the latter does not consider particle-particle interactions.

8:13AM G6.00002 The DC Force Exerted on a Charged Microparticle by an AC Electric Field , DENNIS C. PRIEVE, CHRISTOPHER L. WIRTH, PAUL J. SIDES, Carnegie Mellon University — In 0.15 mM solution of KOH or NaHCO3 in water, a single negatively charged 6-micron polystyrene sphere is levitated about 200 nm above a negatively charged planar ITO electrode by double-layer repulsion. The potential energy profile of forces acting on the sphere is determined by monitoring the distribution of elevations sampled by Brownian motion of the sphere and measured using total internal-reflection microscopy, which can detect changes in elevation as small as 1 nm. Application of a 10 kV/m electric field oscillating at 10 kHz produced oscillations in elevation which were completely swamped by Brownian motion. Nonetheless an unexpected steady attractive force was detected which was comparable in magnitude to the net weight of the sphere (0.05 pN). This additional force was proportional to the square of the electric field amplitude and is about a factor of 2 stronger in KOH compared to NaHCO3. The DC force appears to be dielectrophoretic attraction resulting between aligned dipoles induced in the sphere and the planar electrode by the electric field [Dietz, J. Appl. Phys. 48, 1036 (1977)]. A similar force causes "necklaces" of colloidal particles to form.

8:26AM G6.00003 Analysis of eletrectrohydrodynamic jetting using multifunctional and three-dimensional tomography[1] , HAN SEO KO, XUAN HUNG NGUYEN, SOO-HONG LEE, YOUNG HYUN KIM, Sungkyunkwan University — Three-dimensional optical tomography technique was developed to reconstruct three-dimensional flow fields using a set of two-dimensional shadowgraphic images and normal gray images. From three high speed cameras, which were positioned at an offset angle of 45° relative to one another, number, size and location of electrohydrodynamic jets with respect to the nozzle position were analyzed using shadowgraphic tomography employing a multiplicative algebraic reconstruction technique (MART). Additionally, a flow field inside cone-shaped liquid (Taylor cone) which was induced under electric field was also observed using a simultaneous multiplicative algebraic reconstruction technique (SMART) for reconstructing intensities of particle light and combining with a three-dimensional cross correlation. Various velocity fields of a circulating flow inside the cone-shaped liquid due to different physico-chemical properties of liquid and applied voltages were also investigated.

[1]This work supported by the Basic Science Research Program through the National Research Foundation of Korea (NRF) funded by the Korean government (MEST) (No. S-2011-0023457).

8:39AM G6.00004 Electrokinetic filtration and separation of particles by size in single-spiral microchannels. , JOHN DUBOSE, NATHANIEL TUPPER, JOHN STONAKER, SAURIN PATEL, XIANGCHUN XUAN, Clemson University — In this talk we demonstrate the utility of single-spiral microchannels for the continuous filtration and separation of particles by size. The negative dielectrophoretic force used in manipulating particle trajectories arises from the continual non-uniformity of the imposed electric field within the curved channel. When subjected to an externally imposed direct current power supply to electrokinetically drive the flow, 5, 10, and 15 micrometer polystyrene beads in 1 mM phosphate buffer solutions are independently focused. The various experimentally determined voltages needed for complete particle focusing differ depending upon the diameter of the separate particle species, which allows for the possibility of the continuous filtration and separation of binary particle mixtures at the outlet of the single-spiral microchannel. We also demonstrate an effective separation of a ternary particle mixture by size in a single-spiral microchannel with multiple outlet reservoirs.

8:52AM G6.00005 Ion correlation and ion steric effects on electrophoresis of a colloidal particle , ROBERT STOUT, ADITYA KHAIR, Dept. of Chemical Engineering, Carnegie Mellon University — We revisit the classic problem of electrophoresis of a spherical colloid, using modified PNP equations that account for: (i) steric repulsion between finite sized ions through Bikerman's model [1]; and (ii) electrostatic correlations between ions via a modified Poisson equation recently proposed by Bazant et al. [2]. At low particle zeta potentials, we derive an analytical formula for the electrophoretic mobility accounting for ion correlations, which predicts mobility reversals at sufficiently large ion correlation lengths. Next, we develop an asymptotic scheme for thin Debye layers to compute the mobility for larger zeta potentials, where ion steric effects impose a limit on the counter-ion density in the diffuse Debye layer. Our calculations are compared to experiments on electrophoresis in concentrated multivalent electrolytes.

[1] J. J. Bikerman, Philos. Mag. 33, 384 (1942)
[2] M. Z. Bazant, B. D. Storey, and A. A. Kornyshev, Phys. Rev. Lett. 106, 046102 (2011).

9:05AM G6.00006 Frequency dispersion in dipolophoresis of metallodielectric Janus spheres , ALICIA BOYMELGREEN, GILAD YOSSIFON, Technion - Israel Institute of Technology, TOUVIA MILOH, University of Tel-Aviv — Dipolophoresis (DIP) is an umbrella term for the two non-linear electrokinetic phenomenon of induced-charge electrophoresis (ICEP) and dielectrophoresis (DEP). It has previously been shown that this effect is responsible for the obtainment of a finite velocity by a metallodielectric (comprised of one conducting and one dielectric hemisphere) Janus spheres, even under the application of a uniform AC field. At low frequencies, this mobility is dominated by induced-charge effects, wherein the stronger induced-charge electroosmotic flow around the polarizable hemisphere propels the particle perpendicular to the electric field in the direction of its dielectric end. Surprisingly, it was observed that this motion is at a maximum for applied frequencies in the range of 1kHz beyond which the effect decays. Here we examine the effect of varying experimental conditions including electrolyte concentration and particle size on this limit. Additionally, we present for the first time an analytical solution which is capable of predicting this optimum based on our previous formulation which is uniquely valid for arbitrary electric double layer length. This work is of both fundamental and practical importance and may be used to optimize the behavior of Janus micromotors in lab-on-a-chip systems.

9:18AM G6.00007 Bifurcation in the equililbrium height of colloidal particles over an electrode in low frequency electric fields , TAYLOR WOEHL, Dept. Chemical Engineering & Materials Science, University of California, Davis, CARI DUTCHER, Dept. Mechanical Engineering, University of Minnesota, NICHOLAS TALKEN, BING JIE CHEN, WILLIAM RISTENPART, Dept. Chemical Engineering & Materials Science, University of California, Davis — Colloidal particles are known to change their equilibrium height above an electrode in response to an applied AC electric field, partially due to a lift force caused by electrohydrodynamic (EHD) flow generated around each particle. Here we report the existence of an unexpected bifurcation in the equilibrium particle height in response to low frequency ($\sim$100 Hz) fields. Optical and confocal microscopy observations reveal that upon application of the field 40% of the particles rapidly move several particle diameters up from the electrode, while the remaining 60% move slightly down. Statistics compiled from repeated trials demonstrate that the probability of any particle moving up follows a binomial distribution, indicating that particle lift up is random and does not result from membership in a distinct subpopulation of particles. The observations provide strong evidence for the existence of a tertiary minimum in the interaction potential at a surprisingly large distance from the electrode. We present scaling arguments for the interaction potential in terms a balance between colloidal forces, EHD flow, dipole image attraction, and gravity, yielding a predicted interaction potential with a tertiary minimum that is qualitatively consistent with the observed bifurcation.

9:31AM G6.00008 Nonlinear electrophoresis of ideally polarizable particles , BRUNO FIGLIUZZI, WAI HONG RONALD CHAN, CULLEN R. BUIE, Massachusetts Institute of Technology — We focus in this presentation on the nonlinear electrophoresis of ideally polarizable particles. At high applied voltages, significant ionic exchanges occur between the EDL which surrounders the particle and the bulk solution. In this situation, the velocity field, the electric potential and the ionic concentration at the immediate vicinity of the particle are described by a complicated set of coupled nonlinear partial differential equations. These equations are classically considered in the limit of a weak applied field, which enables further analytical progress (Khair and Squires, Phys. Fluids, 2010). However, in the general case, the equation governing the electrophoretic motion of the particle must be solved numerically. In this study, we rely on a numerical approach to determine the electric potential, ionic concentration and velocity field in the bulk solution surrounding the particle. The numerical simulations use a pseudo-spectral which was used successfully by Chu and Bazant to determine the electric potential and the ionic concentration around an ideally polarizable metallic sphere (Physical Review E, 2006). Our numerical model also incorporates the steric model developed by Kilic *et al.* in 2007 to account for crowding effects in the electric double layer.

9:44AM G6.00009 Thermal dielectrophoretic force on a dielectric particle , BARUKYAH SHAPARENKO, HOWARD HU, HAIM BAU, University of Pennsylvania — A particle immersed in a fluid subjected simultaneously to electric and thermal fields experiences an electrostatic force given by not only classical dielectrophoresis (DEP), but also an additional force, which we term thermal DEP. Assuming the change in the background electric field across the particle and the relative change of temperature-dependent electric properties across the particle are both small, we develop a linearized model to solve the electric field analytically and integrate the Maxwell stress tensor to find an expression for the thermal DEP force for aligned electric and thermal fields. This thermal DEP force is proportional to the temperature gradient, the square of the electric field strength, and the particle's volume. We compute the fully-coupled system in COMSOL to determine a range of validity for our linearized model and show a practical way to superimpose the classical DEP and thermal DEP forces to find the total electrostatic force on the particle relative to the fluid. Additionally, we examine the thermal DEP force and torque on the particle for unaligned fields. Due to the high electrical conductivity of common biological buffers, the thermal DEP force can play an important role when an electric field is used to control and manipulate cells or bacteria.

9:57AM G6.00010 Deterministic separation of particles by electrophoresis: e-DLD , SRINIVAS HANASOGE, RAGHAVENDRA DEVENDRA, FRANCISCO J. DIEZ, GERMAN DRAZER, Department of Mechanical and Aerospace Engineering, Rutgers, The State University of New Jersey, Piscataway NJ 08854 — A suspension of particles of different size driven through an array of posts, by either gravity or flow, results into streams of particles moving in different directions, which can be used for the continuous separation of different species. In this work, we explore a spatially uniform electric field as a driving force. Specifically, we apply an electric field to a quiescent aqueous suspension (constant pH) of spherical particles of different size, across an isotropic two dimensional array of cylindrical posts at different angles with respect to the principal directions of the array and track the motion of particles as they move through the array. In general, the results are in agreement with the existence of deterministic lateral displacement and directional locking, as with the other driving fields. We discuss characterization and separation experiments as well as the advantages of using electric field to separate particles.

Monday, November 25, 2013 8:00AM - 9:57AM –

Session G7 Microfluids: Electro/Magnetic Manipulation 329 - Paulo Arratia, University of Pennsylvania

8:00AM G7.00001 Electrical manipulation of submicron particles by reservoir-based dielectrophoresis (rDEP). , HERBERT HARRISON, MARK JOHNSON, SAURIN PATEL, XIANGCHUN XUAN, Clemson University — Reservoir-based dielectrophoresis (rDEP) is a recently developed technique by our group that exploits the inherent electric field gradients at a reservoir-microchannel junction for particle and cell manipulations. It is based on the induced negative dielectrophoretic motion at the junction to focus, trap and sort micron particles and cells. In this talk we present the experimental and numerical results for the electrical manipulation of submicron particles by rDEP. We study and compare the transport of submicron particles from reservoir to microchannel with negative and positive rDEP, respectively. The effects of electric field and medium concentration are examined. The goal is to implement a continuous particle sorting by negative and positive rDEP.

8:13AM G7.00002 Self-assembly and manipulation of particles on drop surfaces , M. JANJUA, American University in Dubai, I.S. FISCHER, P. SINGH, NJIT — We have recently shown that particles adsorbed on the surface of a drop can be self-assembled at the poles or the equator of the drop by applying a uniform electric field, and that this method can be used to separate on the surface of a drop particles experiencing positive dielectrophoresis from those experiencing negative dielectrophoresis. In this talk we show that the frequency of the electric field is an important parameter which can be used to modify the distribution of self-assembled monolayers.

8:26AM G7.00003 Mathematical modeling of the motion of soft biological particles during insulator based dielectrophoresis , NAGA NEEHAR DINGARI, CULLEN R. BUIE, Massachusetts Institute of Technology — We present a theoretical model to investigate the effects of soft polyelectrolyte layers and bulk ionic concentration on the motion of biological particles such as bacteria in insulator based dielectrophoresis (iDEP) [1] devices. The polarizabilities and electrophoretic mobilities are calculated by solving modified Poisson-Nernst-Plank equations [2] (for ionic transport) and modified Stokes equations (for fluid flow) around the soft particle. The details of soft layer are modeled by including dissociation of ionogenic groups within soft layer and specific interactions with the background electrolyte. We consider two test cases: fibrillated and unfibrillated bacteria whose mobilities were analyzed theoretically and experimentally by Duval et.*al* [3]. We consider a wide range of bulk electrolyte concentration to include thin and thick double layer cases. As a consequence of our analysis we highlight an interesting interplay between soft layer conductivity (function of pH, pKa of ionogenic groups) and double layer conductivity (function of bulk electrolyte concentration) on the particle trajectories in an iDEP device.

[1] Braff, W. a; Pignier, A.; Buie, C. R. *Lab on a chip* **2012**, *12*, 1327–31.
[2] Duval, J. F. L.; Ohshima, H. *Langmuir* **2006**, *22*, 3533–46.
[3] Duval, J. F. L.; Busscher, H. J.; van de Belt-Gritter, B.; van der Mei, H. C.; Norde, W. *Langmuir* **2005**, *21*, 11268–82.

8:39AM G7.00004 Electric filed induced self-assembly of monolayers of sub-micron sized particles on flexible thin films , K. SHAH, M. HOSSAIN, NJIT, M. JANJUA, American University in Dubai, N. AUBRY, Northeastern, I.S. FISCHER, P. SINGH, NJIT — We present a technique that uses an electric field in the direction normal to the interface for self-assembling particle monolayers of sub-micron sized particles on fluid-liquid interfaces and freezing these monolayers onto the surface of a flexible thin film. The electric field gives rise to dipole-dipole and capillary forces which cause the particles to arrange in a triangular pattern. The technique involves assembling the monolayer on the interface between a UV-curable resin and another fluid by applying an electric field, and then curing the resin by applying UV light. The monolayer becomes embedded on the surface of the solidified resin film.

8:52AM G7.00005 Electrical trapping and sorting of particles in an asymmetric ratchet microchannel , AKSHAY KALE, XINYU LU, XIANGCHUN XUAN, Clemson University — Ratchet microchannels have been demonstrated to implement the electrical focusing, trapping, and sorting of various particles. However, no work has thus far been done on the effects of the structure of ratchets on the transport and manipulation of particles in ratchet microchannels. In this talk we present our recent results of the electrokinetic particle trapping and sorting in an asymmetric ratchet microchannel. The ratchet effects (i.e., forward or backward motion) on the trapping effectiveness and location of particles are examined under various DC-biased AC electric fields. The discrepancies in the trapping voltage and location are utilized to demonstrate a continuous electrical sorting of particles by size.

9:05AM G7.00006 Transport of microspheres across liquid-liquid interfaces , STEFFEN HARDT, ASHOK SINHA, Center of Smart Interfaces, TU Darmstadt, Germany, AMLAN MOLLAH, RANJAN GANGULY, Department of Power Engineering, Jadavpur University, India — Experiments with magnetic microspheres crossing the interface between two immiscible polymer solutions under the influence of a magnetic field are reported. The liquids form a bilaminated configuration in a microchannel, allowing a detailed inspection of the liquid-liquid interface. The trajectories of the particles close to the interface are examined using bright-field microscopy and a high-speed camera. During the interaction phase the interface gets deformed and the particles "snap in," indicating that a three-phase contact line is formed. The dependence of the particle-interface interaction on the size of the microspheres is studied, showing that via transfer across a liquid-liquid interface a size separation of particles can be achieved. Comparing the results for 1.29 micron diameter spheres with those for 4.69 micron spheres, it is found that the small particles are able to cross the interface more easily than what is expected from a simple scaling analysis taking into account the balance between magnetic and interfacial forces on the particles. The most likely explanation for this phenomenon involves the line tension that destabilizes smaller particles adsorbed to a liquid-liquid interface more than larger particles.

9:18AM G7.00007 Self-assembled magnetocapillary swimmers , MAXIME HUBERT, GEOFFROY LUMAY, FLORIANE WEYER, NORIKO OBARA, NICOLAS VANDEWALLE, GRASP, University of Liege, B4000 Liege, Belgium — Capillary driven self-assembly consists of suspending small objects at a water-air interface. Due to the effects of wetting, gravity and surface tension, the interface is slightly deformed, inducing a net force between the particles. In the experiments we present, we consider the presence of a vertical magnetic field acting on soft-ferromagnetic particles. Dipole-dipole repulsion competes with capillary attraction such that 2d ordered structures are self-assembling. By adding a secondary horizontal and oscillating magnetic field, periodic deformations of the assembly are induced. Pulsating particle arrangements start to swim, either translating or rotating. The physical mechanisms and geometrical ingredients behind this cooperative locomotion are identified. Furthermore, strategies to control the swimming dynamics are proposed.

9:31AM G7.00008 Structure and dynamics of self-assembling colloidal monolayers in oscillating magnetic fields , ALISON KOSER, PAULO ARRATIA, University of Pennsyvlania — Many fascinating phenomena such as large-scale collective flows, enhanced fluid mixing and pattern formation have been observed in so-called active fluids, which are composed of particles that can absorb energy and dissipate it into the fluid medium. For active particles immersed in liquids, fluid-mediated viscous stresses can play an important role on the emergence of collective behavior. Here, we experimentally investigate their role in the dynamics of self-assembling magnetically-driven colloidal particles which can form highly organized hexagonal lattices that span length scales much larger than a particle diameter. We find that viscous stresses reduce hexagonal ordering, generate smaller clusters, and significantly decrease down the rate of cluster formation, all while holding the system at constant number density. Furthermore, we show that time and length scales of cluster formation depend on and scale with the Mason number (Mn) or ratio of viscous to magnetic forces. Our results suggest that viscous stresses hinder collective behavior in a self-assembling colloidal system. This work is supported by the Army Research Office through the award W911NF-11-1-0488.

9:44AM G7.00009 Models of particle capture in high gradient magnetic separation , ALMUT EISENTRAEGER, IAN GRIFFITHS, DOMINIC VELLA, University of Oxford — High gradient magnetic separation is an efficient way to remove magnetic and paramagnetic particles, such as heavy metals, from waste water. As the water flows through a mesh of magnetized steel wool, high magnetic gradients around the wires attract and capture the particles. We model such a system by considering a single point dipole travelling through a periodic array of magnetized cylinders. We show that there is a critical Mason number (dimensionless flow velocity) below which the particle is captured independent of its initial position. Above this threshold, particle capture is only partially successful and depends on the particle's initial position. We determine the relationship between the critical Mason number and geometry using numerical and asymptotical calculations. To develop these ideas further, we also discuss briefly the aggregation of particles into chains.

Monday, November 25, 2013 8:00AM - 10:10AM –

Session G8 Particle-Laden Flows IV: General Topics 330 - Dimitrios V. Papavassiliou, University of Oklahoma

8:00AM G8.00001 Effects of near-wall turbulence structure on particles of different Schmidt number , QUOC NGUYEN, CHIRANTH SRINIVASAN, DIMITRIOS PAPAVASSILIOU, The University of Oklahoma — The simulation of the trajectories of scalar particles with different Schmidt numbers, Sc, in turbulent channel flow shows that the effects of near-wall turbulence can lead to significantly different transport behavior for different Sc particles. The reason is that different parts and different scales of the coherent near-wall structures contribute to the transport of particles of different Sc. When particles enter the flow field from a common location at the channel wall, these interactions lead to the observation of different concentrations of particles downstream from the source, depending on their Sc and on time. While this is expected based on intuition, it is a rather interesting finding because it opens the possibility to separate particle dispersions according to their Sc using turbulence. A minimum difference in Sc is required, so that distinct transport mechanisms contribute to the transfer of each type of particle. Results from direct numerical simulation at friction Reynolds number of 300 and 600 will be discussed and for Sc that covers five orders of magnitude.

8:13AM G8.00002 Hydrodynamic forces between colliding spheres during mechanical contact , JULIAN SIMEONOV, Naval Research Laboratory — The time-dependent Stokes equations are solved in the gap of O(1 mm) colliding spheres to determine the rate of change of the lubrication forces after the onset of mechanical contact and large deceleration. Mechanical contact is assumed to begin when the gap clearance becomes equal to the size of the O(0.1 micron) micro-asperities present on the surface of real particles. Fourier expansion is used to solve the initial value problem. Assuming small gap clearances, the leading order asymptotic solution is obtained using singular perturbation expansion methods to match the viscous gap solution and the outer inviscid solution. The asymptotic solution provides the dependence of the resistance, added mass and history forces on the sphere velocity, sphere acceleration, the micro-asperity size and the ratio of the sphere diameters. The analytical results can be used to improve the modeling of hydrodynamic forces during mechanical contact in simulations of particle-laden flow or acoustic propagation in fully saturated sediments.

8:26AM G8.00003 Eulerian-Lagrangian large eddy simulations of dense liquid-solid slurry flow through a horizontal pipe , SUNIL AROLLA, JESSE CAPECELATRO, OLIVIER DESJARDINS, Cornell University — A high-fidelity large eddy simulation based Eulerian-Lagrangian methodology is used to investigate the detailed dynamics of liquid-solid slurries in a horizontal pipe. A dynamic Smagorinsky model based on Lagrangian averaging is employed to account for the sub-grid scale effects in the liquid phase. A fully conservative immersed boundary method is used to account for the pipe geometry on a uniform cartesian grid. The liquid and solid phases are coupled through volume fraction and the momentum exchange terms. Particle-particle and particle-wall collisions are modeled using a soft-sphere approach. Mean particle concentration and velocity profiles are computed, showing excellent agreement with experimental data. Covariance statistics are extracted and compared against multiphase turbulence models in the literature. When the bulk liquid velocity is below the critical deposition velocity, particles form a static bed at the bottom that exhibits strong size segregation. Based on our numerical simulations, a critical value for the Froude number is proposed below which the solid particles starts depositing.

8:39AM G8.00004 Decoupling the effects of the streamline curvature and the vorticity on the hydrodynamic forces acting on a spherical particle in rotating flows , TOSHIAKI FUKADA, SHINTARO TAKEUCHI, TAKEO KAJISHIMA, Department of Mechanical Engineering, Osaka University, Japan — Understanding fluid-particle interactions in vortical flows is important for predicting and controlling particle-laden flows. In the present study, the angular velocity and the lift force on the particle in a free vortex (irrotational flow) and a forced vortex (rigidly-rotating flow) are studied by numerical simulation to see the effects of the streamline curvature and the vorticity of the background flows. Based on the non-inertial frame of reference fixed at the particle center, the streamline curvature and particle Reynolds number of the background flows are varied. An original convective boundary condition is proposed for the curved background flows. For both free and forced vortices, the angular velocity of the particle shows self-similar profile with respect to the streamline curvature of the background flow, and the angular velocity is decomposed into two independent contributions of the streamline curvature and the vorticity. As for the lift coefficient, which also exhibits self-similarity with respect to the streamline curvature, the contributions of the streamline curvature, the vorticity and the angular velocity of the particle are decoupled, and a unified correlation equation for both vortices is proposed.

8:52AM G8.00005 Particle interaction in oscillatory Couette and Poiseuille flows[1] , NIMA FATHI, The University of New Mexico, MARC INGBER, University of Colorado Denver, PETER VOROBIEFF, The University of New Mexico — In oscillating Poiseuille flows of relatively dense suspensions, the direction of particle migration changes with the amplitude of oscillation. High amplitudes produce migration toward low shear rate regions of the flow, and vice versa, low oscillation amplitude results in particle migration toward the high shear rate region. We demonstrate that a similar behavior can be observed in a two-particle system, where it can be physically interpreted more easily, and discuss numerical modeling and experimental studies of oscillatory Poiseuille and Couette flows.

[1]This research is supported by the National Science Foundation and (in part) by a gift from the Procter & Gamble Company.

9:05AM G8.00006 On the simulation of turbulent particle-laden flow subject to radiation: Comparison between Eulerian and Lagrangian approaches , AYMERIC VIE, HADI POURANSARI, REMI ZAMANSKY, ALI MANI, CTR Stanford — The objective of this work is to assess the range of applicability of Eulerian particle transport solvers for radiatively driven particle-laden flows with applications in particle solar receivers and cloud dynamics. In particular we consider a triply periodic flow laden with particles subject to homogeneous radiation [studied by Zamansky et. al. 2013]. Heat transfer from particle clusters to the carrier gas generates buoyancy effects, which leads to vorticity generation in the carrier phase. The vortical structures induce preferential concentration and cluster modification. This feedback dynamics leads to a self-sustained state of turbulence. We present numerical investigation of this configuration using both Lagrangian particle models and Eulerian moment methods (EMM). For the Eulerian moment method, the particle density and momentum are solved using different numerical schemes under the monokinetic assumption. We compare the results obtained by both approaches varying the Stokes number, the particle loading as well as the mesh refinement.

9:18AM G8.00007 Radiative heating of a turbulent particle-laden flow: Effects of radiation regimes on turbulence dynamics , ARI FRANKEL, HADI POURANSARI, GIANLUCA IACCARINO, ALI MANI, Department of Mechanical Engineering, Stanford University — Radiation transport modeling has become increasingly important in the design and analysis of advanced thermal-fluid systems such as particle solar receivers. However, the mechanism for the two-way coupling of radiation transport with turbulence and particle dynamics has not been explored. In this work we employed algebraic and differential radiation models in direct numerical simulations of turbulence particle-lade flows subject to external radiative sources. It is shown that different radiation regimes, from optically thin to opaque, lead to significantly different turbulence structures and particle aggregation.

9:31AM G8.00008 Characterization of the temporal evolution of the particle clustering in radiation-induced turbulence , RÉMI ZAMANSKY, ALI MANI, CTR - Stanford University — In the context of particles laden flow, we explore a regime in which turbulence is sustained solely by the radiative energy absorbed by the dispersed phase. Under such conditions, the non-uniformities in particle distribution produce local temperature fluctuations. In response, the resulting buoyancy induced vortical fluid motions alter the particles concentration. From numerical simulations it has been shown that the feedback loop between the local particle concentration, the temperature fluctuations and the convective motion can create and sustain turbulence in a wide range of parameters, whose key parameter is the particle response time. In particular the temperature variance as well as the turbulent kinetic energy present a sharp peak for maximum particle clustering. The time scales of the temperature and momentum forcing are therefore highly influenced by the "clusters life time." We employed a method that enables to track the temporal evolution of clusters and detect the clusters merging and splitting. This approach uses the Voronoi tessellation of the particle positions (and the connectivity of its cells) to detect the clusters. By introducing a cluster-cluster correlation we construct a random graph representative of the cluster temporal evolution.

9:44AM G8.00009 A low-Mach approximation computational framework for particle-laden flows subject to radiation , HADI POURANSARI, Department of Mechanical Engineering, Stanford University, REMI ZAMANSKY, Center for Turbulence Research, ALI MANI, Department of Mechanical Engineering, Stanford University — The three-way coupled physics of radiation, fluid flow, and particle transport forms the dynamical ingredients in various technological and natural systems, such as particle-based solar-thermal systems, clouds, soothing flames, and atmospheric aerosols. Depending on radiation intensity, the density fluctuations in such systems can be up to order of the mean density itself. We present a parameterization of this problem using a simple model considering flow laden with particles with finite momentum relaxation time. We further present a coupled computational algorithm for simulation of flow, particle transport, and heat transfer using low-Mach approximation. Variety of statistics for gas and dispersed phases are investigated to depict the effect of radiation on particle-laden turbulence at different scenarios. The range of applicability of Boussinesq approximation for modeling buoyancy effects will be discussed.

9:57AM G8.00010 Large-Eddy Simulation of Particle Dispersion Inside and Above Plant Canopies , YING PAN, MARCELO CHAMECKI, SCOTT ISARD, Pennsylvania State University — Modeling the dispersion of small particles such as pathogenic spores, pollens, and small seeds inside and above plant canopies is important in many applications. Transport of these particles is driven by strongly inhomogeneous, coherent, and non-Gaussian turbulent flows inside the canopy roughness sublayer (the regions extending from ground to about three canopy heights). We develop an LES approach that includes parameterization of plant reconfiguration through a velocity-dependent drag coefficient and yield predictions of turbulence statistics and coherent structures in good agreement with experimental data. Particle dispersion is also validated against experimental data of spore dispersal inside and above a maize field. LES results are used in the development of a simple framework for modeling the particle plume. Characteristics of the particle plume in the near and far fields are studied. Results suggest that the far field plume can be approximated by a simple analytical solution if the fraction of spores that escape the canopy region is known.

Monday, November 25, 2013 8:00AM - 10:10AM –

Session G9 Instability: Interfacial and Thin-Film III 333 - Paul Steen, Cornell University

8:00AM G9.00001 Dynamics of a thin ferrofluid film subjected to a magnetic field , DEVIN CONROY, ALEX WRAY, OMAR MATAR, Imperial College London — We consider a thin film flowing down a rigid, impermeable inclined plane subjected to a magnetic field. The film corresponds to a ferrofluid and is bounded from above by a hydrodynamically-passive gas. The ferrofluid is considered to be weakly-conducting, and its dynamics are governed by the steady Maxwell's equations, coupled to the Navier-Stokes, and continuity equations. The magnetisation of the ferrofluid is a function of the magnetic field, which can be represented by a nonlinear Langevin function. We use long-wave theory to determine the velocity and pressure fields in the film, and use Fourier transforms to solve for the potential field in the gas phase. Application of the interfacial and kinematic boundary conditions then leads to a one-dimensional partial differential equation for the interface with a non-local contribution from the magnetic effects. A linear stability analysis of this equation is carried out. This equation is then solved numerically for a wide range of system parameters. The results of this parametric study will be presented.

8:13AM G9.00002 Transition to disorder: the effects of elasticity on thin film flow inside a tube[1] , JEFFREY OLANDER, ROBERTO CAMASSA, G.M. FOREST, H. REED OGROSKY[2], University of North Carolina — Previous studies of non-Newtonian flows driven up a tube by a high volume flux flow of air have suggested that a transition to disordered core-annular wave dynamics occurs when the liquid becomes elastic. Understanding this transition may shed light on the behavior of mucus in the human trachea. We present results from experiments of thin-film liquid flows of a Newtonian fluid and a non-Newtonian, dilute mixture of oligomers. These so-called Boger fluids are elastic, non-thixotropic liquid solutions made by dissolving a non-Newtonian solute in a Newtonian base. We compare, through video analysis, the wave dynamics of the Boger fluid to those of its Newtonian base under identical inflow conditions. We describe observed differences between the Newtonian and non-Newtonian cases. Finally, quantitative comparisons of wave properties and liquid mass transport are discussed.

[1]We would like to thank the National Science Foundation (DMS grants 0509423, 1009750, RTG -0943851) and the National Institutes of Health (NIEHS 534197-3411) for supporting this study

[2]Currently a postdoctoral fellow at the University of Wisconsin - Madison

8:26AM G9.00003 Laminar flow over a thin film , THOMAS WARD, PAUL TROUPE, Iowa State University — Thin film deformation that is driven by an external laminar flow over a flat plate is considered using both theoretical and computational analysis. To perform theoretical analysis we utilize the Blasius boundary layer solution to develop the thin film evolution equation. Non-dimensionalization of the resulting film evolution equations yields two dimensionless parameters, the Weber and Reynolds numbers. The film thickness is computed for a wide range of both Weber (> 1) and Reynolds (> 1) numbers using standard disjoining pressure models. The scaling and computed results suggest strong dependence on the Reynolds number, where in the limit of large Reynolds number the evolution equations are self similar.

8:39AM G9.00004 Buckling of a thin, viscous film in an axisymmetric geometry[1] , MORRIS FLYNN, SANJAY BHATTACHARYA, Univ. of Alberta, Dept. of Mech. Eng., RICHARD CRASTER, Imperial College London, Dept. of Mathematics — By adapting the Föppl-von Kàrmàn equation, which describes the deformation of an elastic membrane, we consider the buckling pattern of a thin, very viscous fluid layer subject to shear in an axisymmetric geometry. A linear stability analysis yields a differential eigenvalue problem, whose solution by spectral methods, yields the most unstable azimuthal wave-number, m^*. Contrary to the discussion of Slim et al. (*J. Fluid Mech.*, **694**, pp. 5-28, 2012), we argue that the axisymmetric problem shares the same degeneracy as its rectilinear counterpart so that at the onset of instability, m^* is indefinitely large. Away from this point, however, a comparison with analogue experimental measurements is both possible and generally favorable. In this vein, we describe the laboratory apparatus used to make new measurements of m^*, the phase speed and the wave amplitude; no prediction concerning the latter two quantities can be made using the present (self-adjoint) theory. Experiments reveal a limited range of angular velocities where waves of either small or large amplitude may be excited. In contrast to the analogue problem from solid mechanics, transition from one to the other regime does not appear to be associated with a notable change in m^*.

[1]Funding provided by NSERC

8:52AM G9.00005 Vortices catapult droplets in atomization , J. JOHN SOUNDAR JEROME, Institut D'Alembert (IJLRDA), CNRS - UPMC, France, SYLVAIN MARTY, JEAN-PHILIPPE MATAS, Laboratoire des Écoulements Géophysique et Industriels (LEGI), CNRS - Université de Grenoble, France, STEPHANE ZALESKI, JEROME HOEPFFNER, Institut D'Alembert (IJLRDA), CNRS - UPMC, France — A droplet ejection mechanism in planar two-phase mixing layers is examined. Any disturbance on the gas-liquid interface grows into a Kelvin-Helmholtz wave and the wave crest forms a thin liquid film that flaps as the wave grows downstream. Increasing the gas speed, it is observed that the film breaks-up into droplets which are eventually thrown into the gas stream at large angles. In a flow where most of the momentum is in the horizontal direction, it is surprising to observe these large ejection angles. Our experiments and simulations show that a recirculation region grows downstream of the wave and leads to vortex shedding similar to the wake of a backward-facing step. The ejection mechanism results from the interaction between the liquid film and the vortex shedding sequence: a recirculation zone appears in the wake of the wave and a liquid film emerges from the wave crest; the recirculation region detaches into a vortex and the gas flow over the wave momentarily reattaches due to the departure of the vortex; this reattached flow pushes the liquid film down; by now, a new recirculation vortex is being created in the wake of the wave–just where the liquid film is now located; the liquid film is blown-up from below by the newly formed recirculation vortex in a manner similar to a bag-breakup event.

9:05AM G9.00006 Collapse Dynamics in Rotating Thin Films , SHOMEEK MUKHOPADHYAY, Levich Institute, City College of New York, JOSHUA DIJKSMAN, Physics Department, Duke University, RICHARD MACLAUGHLIN, ROBERTO CAMASSA, Department of Mathematics, University of North Carolina at Chapel Hill, ROBERT BEHRINGER, Physics Department, Duke University — We study the collapse of a dry cavity in a thin fluid film. Collapse dynamics driven by viscous gravity currents is well described by a self similar solution of the thin film equation. In the current analysis we include surface tension effects. We find both experimentally and numerically that for small capillary numbers, the collapse dynamics retains its power law scaling behavior, but with a power law clearly distinct from viscous collapse dynamics.

9:18AM G9.00007 ABSTRACT WITHDRAWN –

9:31AM G9.00008 Wrinkling of Thin Films Induced by Viscous Stress , SOURAV CHATTERJEE, CHRISTINA MCDONALD, JIANI NIU, University of Pittsburgh, RUI HUANG, University of Texas, Austin, SACHIN VELANKAR, University of Pittsburgh — Compression of thin films attached to compliant solid substrates can induce a variety of highly ordered and complex wrinkling patterns. We study an analogous problem of the wrinkling instability of a thin film floating on a viscous fluid. Uniaxial compression of the fluid induces a viscous stress which leads to the wrinkling of the film. We experimentally determine the effect of geometry and material properties on the wrinkle wavelength. A shear lag approach is used to determine the stress distribution prior to buckling. A linear stability analysis of the film under this stress distribution is used to determine the maximally growing wavelength in the system. Both experiments as well as stability analysis show that the wavelength depends significantly on film length and the ratio of the film and fluid layer thickness. Most importantly, unlike previous research on fluid-supported films, the wrinkle wavelength is rate-dependent, and reduces with increasing compression rate.

9:44AM G9.00009 Meniscus stability in the planar-flow melt spinning of thin metallic sheets , ANTHONY ALTIERI, PAUL STEEN, Cornell University — Planar-flow melt spinning is a process for the continuous fabrication of thin, metallic sheets or ribbons. During solidification, molten metal-air meniscus vibrations create periodic thickness variations on the final product. It has been observed that flow configuration and contact line conditions affect the frequency of thickness variations. A model problem of a free interface enclosing an inviscid fluid in a slot is examined. A linear stability analysis shows the effect of flow near the interface for various contact line constraints.

9:57AM G9.00010 Experimental investigation of the stability of a moving radial liquid sheet[1] , MANJULA PARAMATI, MAHESH TIRUMKUDULU, Department of Chemical Engineering, Indian Institute of Technology Bombay, Powai, Mumbai - 400076. India — Experiments were conducted to understand the stability of moving radial liquid sheets formed by the head-on impingement of two co-linear water jets using laser induced fluorescence technique (LIF). Acoustic sinusoidal fluctuations were introduced at the jet impingement point and we measured the displacement of the center line of the liquid sheet (sinuous mode) and the thickness variation (varicose mode) of the disturbed liquid sheet. Our experiments show that the sinuous disturbances grow as they are convected outward in the radial direction even in the smooth regime (We $<$ 800). In the absence of the acoustic forcing, the measured thickness has the expected 1/r dependence. Interestingly, we were unable to detect any thickness variation about the pre-stimulus values in the presence of acoustic forcing suggesting that the variation in the thickness is lower than the resolution of the technique ($\pm$ 1 μm). The growth rates of the sinuous mode determined from the wave envelope matches with the prediction of a recent theory by Tirumkudulu and Paramati (Communicated to Phys. Of Fluids, 2013) which accounts for the inertia of the liquid phase and the surface tension force in a radial liquid sheet while neglecting the inertial effects due to the surrounding gas phase.

[1]The authors acknowledge the financial assistance from Indo-French Center for Pro- motion of Advanced Research and also Indian institute of technology Bombay.

Monday, November 25, 2013 8:00AM - 9:57AM –

Session G10 Instability: Wakes I - Cylindrical Objects 334 - Paul Fontana, Seattle University

8:00AM G10.00001 Cylinder wakes in quasi-two-dimensional flows with surface friction I: instability and scaling[1]

, JEMIN SHIM, JAMIE H. W. LI, DAVID F. RASCHKO, PAUL W. FONTANA, Seattle University — We measured the frequency of vortex shedding produced by cylinders in a quasi-two-dimensional system with homogenous drag. The system is characterized by the Reynolds number $Re = U_0D/\nu$ (U_0 = flow speed without the obstacle, D = cylinder diameter, ν = kinematic viscosity), and a dimensionless drag parameter, $\alpha^* = D^2/(L_s^2Re)$ (L_s = length scale above which drag force exceeds viscous force). We investigated the scaling of the Strouhal number $St = fD/U_o$ (f = vortex shedding frequency) and compared it with conventional measurements in flows without homogenous drag. The dynamics bifurcates above a critical diameter $D_c \sim L_s$, indicating that the effect of surface friction becomes important. Increased fluctuations beyond the bifurcation indicate the onset of a previously unobserved instability associated with the drag. Also, near some critical parameters, shear instability without vortex shedding is observed, with vortex streets appearing at both higher and lower Reynolds number; the mode at lower Reynolds number has not previously been observed.

[1]Supported by the National Science Foundation under Grant No. CBET-0854509, the M. J. Murdock Charitable Trust, and the Clare Boothe Luce Foundation.

8:13AM G10.00002 Cylinder wakes in quasi-two-dimensional flows with surface friction II: effects of film thickness[1]

, JAMIE H.W. LI, JEMIN SHIM, PAUL W. FONTANA, Seattle University — Vortex shedding in a quasi-two-dimensional system with homogeneous drag (Ekman friction) is observed to have different phenomenology than in systems without friction. To understand why, we studied the wakes of circular cylinders in a vertical soap film channel and measured thickness profiles (pachymetry) of the film in the cylinder wake. The kinematic viscosity and drag coefficients in this system both depend on the thickness of the soap film, which varies over the wake. To measure thickness, broad-spectrum light is reflected off the film, and the resulting interference pattern of intensity vs. wave number is measured. The spacing in wave number of the interference minima is proportional to the film thickness, giving high-accuracy thickness measurements with a precision on the order of 0.2%. Pachymetry profiles transverse to the mean flow were measured at five longitudinal positions for various values of Reynolds number and drag parameter. Possible causes for differences in the dynamics from conventional systems could be: ambiguity in the specifications of Reynolds number or non-Newtonian effects arising from viscosity gradients, elastic effects particular to soap films, or surface friction. The pachymetry results favor the latter explanation.

[1]Supported by the National Science Foundation under Grant No. CBET-0854509, the M. J. Murdock Charitable Trust, and the Clare Boothe Luce Foundation.

8:26AM G10.00003 An iterative methodology for the computation of perturbation fields induced by harmonic forcing of the linearised Navier-Stokes equations in complex geometries and application to forced cylinder wakes[1]

, GEORGE PAPADAKIS, LIANG LU, Imperial College London — An efficient, iterative methodology is developed for the computation of the perturbation fields induced by harmonic forcing of the linearised Navier-Stokes equations in complex geometries. The problem is formulated in the frequency domain and the resulting system of equations is solved iteratively until convergence. This approach offers distinct advantages: convergence is monitored easily, and the solution from one value of frequency can be used as a restart field for another, nearby, frequency. It is also straightforward to implement in any implicit code that solves the steady Navier-Stokes equations iteratively. The method can be extended to solve the optimal forcing problem, i.e. to find the forcing fields that will maximise the energy of the flow perturbations for a particular frequency. In the present study, the method is applied to investigate the wake behind a cylinder with pulsating approaching flow. The perturbation velocity and pressure fields induced by external forcing are computed and the mechanisms that drive the energy growth of the developed structures in the wake are examined. It is shown that perturbations grow by extracting energy from two sources: the underlying base flow field and the externally provided energy that maintains the imposed oscillation.

[1]Part of this work was supported by EPSRC EP/I016015/1

8:39AM G10.00004 A Zoology of unstable modes in a stratified cylinder wake

, MICKAEL BOSCO, PATRICE MEUNIER, None — Although the dynamics of a cylinder wake is well known and extremely rich for a homogeneous fluid, very few studies have been focused on stratified wakes despite the obvious extensive number of applications for geophysical flows and submarine wakes. The presence of the stratification may largely modify the dynamics of the wake. The study is devoted to understand the effect of the tilt and also of a strong stratification. So extensive experimental and numerical results have been investigated to describe the full dynamics of a tilted cylinder wake. For weak stratification and small tilt angle, the classical mode A found for a homogeneous fluid is still present, but for a large tilt angle, an instability appearing far from the cylinder is created. The case of a cylinder towed a very stratified fluid has been finally investigated. The dynamics is strongly modified and for moderate tilt angles, a new unstable mode appears with a structure similar to the Kelvin-Helmholtz billows (observed in the critical layer of a tilted stratified vortex), whereas for large tilt angles, another unstable mode characterized by a strong shear appears generated without a 2D von Karman structure. This reveals the rich dynamics of the cylinder wake in the presence of a stable stratification.

8:52AM G10.00005 ABSTRACT WITHDRAWN –

9:05AM G10.00006 Stability of Flow around a Cylinder in Plane Poiseuille Flow

, HUA-SHU DOU[1], AN-QING BEN[2], Zhejiang Sci-Tech University, FLUID MECHANICS RESEARCH TEAM — Simulation of Navier-Stokes equations is carried out to study the stability of flow around a cylinder in plane Poiseuille flow. The energy gradient method is employed to analyze the mechanism of instability of cylinder wake. The ratio of the channel width to the cylinder diameter is 30, and the Reynolds number based on the cylinder diameter and incoming centerline velocity is 26 and 100, respectively. The incoming flow is given as being laminar. It is found that the instability of the cylinder wake, starting near the front stagnation point upstream. The recirculation zone behind the cylinder has no effect on the stability of the wake. In the wake behind the recirculation zone, the flow stability is controlled by the energy gradient in the shear layer along the two sides of the wake. At high Re, the energy gradient of averaged flow in the channel interacts with the wake vortex, strengthening the wake vortex structure. Due to the large ratio of the channel width to the cylinder diameter, the disturbance caused by the cylinder mainly occurs in the vicinity of the centerline and has little effect on the flow near the wall. The velocity profile on the two sides of the cylinder wake in the downstream channel remains laminar (parabolic profile).

[1]Professor in Fluid Mechanics; AIAA Associate Fellow.
[2]Graduate student in Fluid Mechanics.

9:18AM G10.00007 Too big to grow: the saturation mechanism of the von Karman vortex street captured by a self-consistent model , VLADISLAV MANTIC-LUGO, CRISTOBAL ARRATIA, FRANCOIS GALLAIRE, EPFL-LFMI — The supercritical instability leading to the Karman vortex street in a cylinder wake is a well studied problem: the steady solution becomes linearly unstable and saturates into a limit cycle. However a simple physical picture for understanding the saturation amplitude is still missing. We present a simple self-consistent model that captures the saturation mechanism. The model shows that the main nonlinear effects of the saturation process are retained by the coupling of the mean flow and perturbation equations through the Reynolds stress, which is built only with the first harmonic calculated as the most unstable eigenmode. A simple physical picture is revealed, wherein the perturbation amplitude is such that the modified mean flow is neutrally stable. The mean flow velocity field and the Reynolds stress spatial structure are thus well approximated in a self-consistent manner without any DNS data. Moreover, the results show an accurate vortex shedding frequency prediction when compared to experiments.

9:31AM G10.00008 Investigation of the effect of the spanwise forcing on vortex shedding suppression in the flow past a cylinder[1] , GABRIELE ROCCO, SPENCER SHERWIN, Imperial College London — Controlling the wake vortex dynamics of bluff bodies efficiently is a fundamental problem in many applications. Earlier direct numerical simulations (Darekar, Sherwin, 2001) of three-dimensional bluff bodies have demonstrated that the introduction of a spanwise waviness at both the leading and trailing surfaces suppresses the vortex shedding and reduces the amplitude of the fluctuating aerodynamic forces. Under this motivation, direct numerical simulations and stability analysis of the flow past a three-dimensional cylinder in the supercritical regime were performed. Starting from a fully developed shedding, a sufficiently high spanwise forcing is introduced on the surface of the cylinder, in the regions where separation effects occur, resulting in the stabilisation of the near wake in a time-independent state. Numerical experiments were conducted to detect the critical values of the amplitude of the forcing capable of suppressing the vortex street, and three different physical structures of the wake were detected. Stability analysis of the linearised Navier-Stokes equations was then performed on the three-dimensional flows to investigate the role of the spanwise modulation on the absolute instability associated with the von Kárman street.

[1]EPSRC grant: EP/H050507/1

9:44AM G10.00009 The Wake Analysis Behind a Foamed Cylinder , AMIR KHASHECHI, Azad University of Tehran, RESEARCH CENTER OF AZAD UNIVERSITY OF TEHRAN TEAM — Particle Image Velocimetry (PIV) has been carried out to investigate the wake region behind a foamed and a finned cylinder. The purpose of this analysis is to develop one- and two- point correlations and to investigate the flow characteristics for these two cases. The experiments are conducted for two Reynolds numbers (based on the mean air velocity and the cylinder diameter) 2000 and 8000. The application of Proper Orthogonal Decomposition (POD) to the PIV velocity fields of the two cylinder types is also discussed. The POD computed for the measured velocity fields for both cases shows that the first two spatial modes contain most of the kinetic energy of the flow irrespective to the cylinder type. These two modes are also responsible for the large-scale coherence of the fluctuations. For foamed cylinder types, the first four eigenmodes of the velocity field were measured and their organizations were investigated. These eigenmodes disclose the overall mean flow structure, and the large- scale structure being essentially connected to the most robust flow motion.

Monday, November 25, 2013 8:00AM - 10:10AM –

Session G11 Bubbles IV: Growth, Heat Transfer and Boiling 335 - David Brutin, Polytech Marseille/Centre de la Recherche

8:00AM G11.00001 Electrolytic Bubble Growth on Pillared Arrays[1] , KENNETH LEE, OMER SAVAS, Department of Mechanical Engineering, University of California at Berkeley — In current energy research, artificial photosynthetic (AP) devices are being designed to split water and harvest hydrogen gas using sunlight. In one such design, hydrogen gas bubbles evolve on catalytic surfaces of arrayed micropillars. If these bubbles are not promptly removed from the surface, they can adversely affect gas evolution rates, water flow rates, sunlight capture, and heat management of the system - all of which deteriorate device performance. Therefore, understanding how to remove evolved gas bubbles from the pillar surfaces is crucial. Flow visualization of electrolytic bubble nucleation and detachment from the catalytic pillar surfaces has been conducted. The bubble departure diameter and lift-off frequency are extracted and compared with known correlations from boiling heat transfer. Bubble tracking indicates that bubble detachment is enhanced by local interactions with neighboring bubbles. These observations suggest how hydrogen gas bubbles can be effectively removed from pillared surfaces to prolong AP device longevity.

[1]Joint Center for Artificial Photosynthesis, a U.S. Department of Energy (DOE) Energy Innovations Hub

8:13AM G11.00002 How do bubbles grow in a weakly supersaturated solution? , OSCAR ENRIQUEZ, CHAO SUN, DETLEF LOHSE, University of Twente, ANDREA PROSPERETTI, Johns Hopkins University/University of Twente, DEVARAJ VAN DER MEER, University of Twente — Beer, champagne and soft-drinks are water-based solutions which owe their "bubbliness" to a moderate degree of carbon dioxide supersaturation. Bubbles grow sequentially from nucleation sites due to solute concentration gradients and detach due to buoyancy. The leading mass transfer mechanism is diffusion, but the advection caused by the moving surface also plays an important role. Now, what happens at the limit of very weak supersaturation? We take an experimental look at CO_2 bubbles growing in water under such a condition. Nucleation sites are provided by hydrophobic micro-cavities on a silicon chip, therefore controlling the number and position of bubbles. Although advection is negligible, measured growth rates for an isolated bubble differ noticeably from a purely diffusive theoretical solution. We can explain the differences as effects of the concentration boundary layer around the bubble. Initially, its interaction with the surface on which the bubble grows slows the process down. Later on, the growth rate is enhanced by buoyancy effects caused by the depletion of the solute in the surroundings of the bubble. When neighboring bubbles are brought into play they interact through their boundary layers, further slowing down their growth rates.

8:26AM G11.00003 Mixing and drift by air bubbles crossing an interface of a stratified medium , L. DIAZ-DAMACILLO, A. RUIZ-ANGULO, R. ZENIT, Universidad Nacional Autonoma de Mexico — The dynamics of a single air bubble crossing the horizontal interface separating two different-density stagnant Newtonian miscible liquids are studied experimentally. Both liquids were water-glycerin mixtures. The bottom fluid was saturated with salt to make it denser the upper one. The size of the bubbles was widely varied to obtain a wide range of shapes from spherical to toroidal. The Planar Laser-Induced Flourescence (PLIF) technique was used to quantify the drift volume across the interface. When the bubble crosses the interface, it drags some amount of the heavy fluid into the upper lighter fluid. For small bubbles, the drift volume returns to the bottom liquid after sometime with negligible mixing. The dragged volume is inversely proportional to the bubble Reynolds number. For larger bubbles, the drift volume becomes unstable, which leads to mixing. Considering a balance of inertial, viscous and buoyant forces, we propose a dimensionless number to identify the onset of instability.

8:39AM G11.00004 Effects of wakes and surface contamination on instantaneous mass transfer from a bubble to the surrounding liquid , YOSHINORI NOBATA, Graduate School of Engineering, Shizuoka University, TAKAYUKI SAITO, Research Institute of Green Science and Technology, Shizuoka University — The effects of wakes and surface contamination on instantaneous mass transfer from a zigzagging bubble to the surrounding liquid are discussed. A zigzagging CO2 bubble was captured with high spatial resolution by a pair of high-speed cameras, and the instantaneous changes in volume and surface area of the bubble were obtained from the images by our originally developed method. The instantaneous mass transfer from the bubble to surfactant-contaminated water was reduced, compared with that of the same-size bubble in pure water. In the surfactant-contaminated water, the surface tension of the bubble-liquid interface decreased. In association with the bubble ascent, the adsorbed surfactant was accumulated on the rear of the bubble. This non-uniform distribution of the surfactant on the interface caused the Marangoni flow on the bubble surface. As a result, the Marangoni effect attenuated the bubble motion and the vortex shedding. On the other hand, in pure water the shedding of hairpin vortexes from the bubble rear was very active. The hairpin vortexes transported the CO2-rich liquid away from the bubble. These indicate that the vortex shedding promotes convective transportation of CO2 and induced the enhancement of the instantaneous mass transfer.

8:52AM G11.00005 Using Improved Equation of State to Model Simultaneous Nucleation and Bubble Growth in Thermoplastic Foams , IRFAN KHAN, STEPHANE COSTEUX, DAVID ADRIAN, DIEGO CRISTANCHO, The Dow Chemical Company — Due to environmental regulations carbon-dioxide (CO_2) is increasingly being used to replace traditional blowing agents in thermoplastic foams. CO_2 is dissolved in the polymer matrix under supercritical conditions. In order to predict the effect of process parameters on foam properties using numerical modeling, the P-V-T relationship of the blowing agents should accurately be represented at the supercritical state. Previous studies in the area of foam modeling have all used ideal gas equation of state to predict the behavior of the blowing agent. In this work the Peng-Robinson equation of state is being used to model the blowing agent during its diffusion into the growing bubble. The model is based on the popular "Influence Volume Approach," which assumes a growing boundary layer with depleted blowing agent surrounds each bubble. Classical nucleation theory is used to predict the rate of nucleation of bubbles. By solving the mass balance, momentum balance and species conservation equations for each bubble, the model is capable of predicting average bubble size, bubble size distribution and bulk porosity. The effect of the improved model on the bubble growth and foam properties are discussed.

9:05AM G11.00006 Asymmetric interface temperature during vapor bubble growth , ANTOINE DIANA, Aix-Marseille University, MARTIN CASTILLO, TED STEINBERG, Queensland University of Technology, DAVID BRUTIN, Aix-Marseille University, AMU COLLABORATION, QUT COLLABORATION — We investigate the nucleation, growth, and detachment of single vapor bubbles at the interface microscale. Shear flow is used to investigate pool and convective boiling situations using visible and infrared visualizations. We determine a threshold Reynolds number for the onset of asymmetric interfacial temperatures. Below this threshold, bubble growth is geometrically and thermally symmetric, while above, bubbles no longer grow thermally symmetrically. This is explained by the dominance of convective heat transfer removal over viscous effects at the bubble interface. We experimentally demonstrate asymmetric interfacial temperature profiles that should be taken into account for future bubble growth modeling.

9:18AM G11.00007 Single Bubble Dynamics on Superhydrophilic Micropillar Arrays during Flow Boiling[1] , JIANSHENG FENG, SIYU CHEN, Massachusetts Institute of Technology, TIEJUN ZHANG, Masdar Institute of Science and Technology, EVELYN WANG[2], Massachusetts Institute of Technology, DEVICE RESEARCH LABORATORY TEAM — Micro/nanoengineered surfaces have received recent interest for high heat flux thermal management solutions. In particular, micropillar arrays promise opportunities to enhance flow boiling performance, but an increasing understanding of the role of these structures are still needed. In this study, we used superhydrophilic micropillar arrays with well-defined geometries to investigate bubble growth and departure dynamics during boiling. These structures were individually tested in a closed-loop flow boiling setup. A combined side-view microscopy and high-speed videography technique was utilized to obtain images of bubble growth and departure. We demonstrated that by increasing the solid fraction of the microstructures, bubble departure can occur at smaller sizes and at higher frequencies comparing to that on a flat hydrophilic surface. Meanwhile, we observed that bubble sliding stage between departing from the nucleation site and detaching from the heated surface, which is present under a wide range of conditions during flow boiling on flat surfaces, was highly suppressed on some of the microstructured surfaces. In addition, we used a surface energy based model to explain the confinement effect of the liquid-vapor interface by the micropillar arrays, and to support our experimental findings that solid fraction is a key parameter dictating bubble dynamics.

[1]This work is supported by the Masdar Institute of Science and Technology.
[2]Principal Investigator, corresponding author

9:31AM G11.00008 Dual-Luminescent Imaging for Capturing Temperature Field around a Bubble , HIROTAKA SAKAUE, JAXA, HIDEKI GOYA, TAKESHI MIYAZAKI, The University of Electro-Communications — Dual-luminescent imaging uses two-luminescent outputs to extract the temperature information from an acquired image. This is applied to capture the temperature field around a bubble in boiling water. A laser sheet is used as an illumination source to obtain a temperature profile of the bubble. By using a fast frame-rate camera as an image acquisition unit, the time-resolved temperature information around the bubble can be captured. In the presentation, a current status of this measurement will be presented.

9:44AM G11.00009 Heat transfer in turbulent bubbly flow in channels[1] , SADEGH DABIRI, GRETAR TRYGGVASON, University of Notre Dame — In many applications convective heat transfer occurs in the presence of a multiphase turbulent flow such as in boilers and bubble column reactors. Turbulence in channel and pipe flows significantly increases the heat transfer rate. Here we examine the effect of turbulent bubbly flows on the heat transfer inside a vertical channel with uniform heat flux on the walls and compare it with the heat transfer in single phase flow. Both bubbles and the turbulence are fully resolved through Direct Numerical Simulation. The distribution of the bubbles in the channel is affected by the deformability of the bubbles. A wall-peaked distribution is observed for nearly spherical bubbles and a core-peaked distribution is observed for deformable bubbles. This change in distribution of the bubbles significantly affects the flow rate in the channel and the heat transfer rate as well. The results of heat transfer for different flow configurations are presented and compared to the heat transfer in a single phase channel flow.

[1]This research is partially supported by CASL project.

9:57AM G11.00010 Multi-Scale Acoustic Actuation of Vapor Bubbles for Pool Boiling Enhancement , THOMAS R. BOZIUK, MARC K. SMITH, ARI GLEZER, Georgia Institute of Technology — The effect of multi-scale acoustic actuation on heat transfer from a submerged structured surface in pool boiling is investigated experimentally. Actuation over a range of frequencies affects the growth, detachment, advection, and condensation of vapor bubbles and results in significant favorable changes to the boiling curve and critical heat flux. Heat transfer is also improved with a structured heated surface containing fixed but separate nucleation sites designed to limit the merger of vapor bubbles above the surface and to enable an efficient inflow of makeup liquid to the evaporation sites. However, the geometry of the surface between the evaporation sites can impede the effectiveness of the acoustic actuation within certain bandwidths related to the scale of the geometry. It is shown that a multi-scale approach combining low frequency (kHz-range) actuation, for bubble interface excitation and enhanced condensation, with high frequency (MHz-range) actuation, for induced interfacial forces near the contact line, yields effective control of the evolution of vapor bubbles over a broad range of scales and surface geometries and leads to a significant improvement in boiling heat transfer.

Monday, November 25, 2013 8:00AM - 10:10AM –

Session G12 Vortex Dynamics and Vortex Flows IV 336 - Melissa Green, Syracuse University

8:00AM G12.00001 Investigating three-dimensional wake topology of a low aspect ratio dual step cylinder with 2D PIV measurements , CHRIS MORTON, SERHIY YARUSEVYCH, University of Waterloo — A dual step cylinder is composed of a large diameter cylinder (D) of small aspect ratio (L/D) attached to the mid-span of a small diameter cylinder (d). The present work investigates the flow past dual step cylinders for $Re_D = 2100$, $0.2 \leq L/D \leq 3$, and $1.33 \leq D/d \leq 2.67$. Experiments are completed in a water flume facility employing Laser Doppler Velocimetry (LDV) and planar Particle Image Velocimetry (PIV), as well as hydrogen bubble flow visualization. Turbulent vortex shedding occurs in the wake of the dual step cylinder for all the cases investigated. However, wake topology and vortex dynamics are influenced significantly by the geometrical parameters of the model, namely, L/D and D/d. A novel method is introduced for reconstructing salient features of the three-dimensional wake topology using phase-averaged 2D PIV measurements. The results show that flow development in the small cylinder wake away from the large cylinder is similar to that expected for a uniform cylinder of the same diameter. However, complex three-dimensional vortex deformations and splitting occur downstream of the large diameter cylinder. Four distinct flow regimes are identified based on changes in large cylinder wake development: (i) vortex shedding at a frequency lower than that expected for a uniform cylinder, (ii) irregular shedding, (iii) vortex shedding at a frequency higher than that for a uniform cylinder, and (iv) suppression of large cylinder vortex shedding.

8:13AM G12.00002 Numerical investigation of vortex shedding behind a square cylinder oscillating in a closed channel , JOHN PITRE, JOSEPH BULL, University of Michigan — We investigate the vortex shedding behavior of a square cylinder oscillating longitudinally in a closed channel at a high blockage ratio (>80%). The square cylinder translates parallel to the long axis of the channel with a sinusoidal velocity of a given oscillation frequency. The incompressible Navier-Stokes equations in the Arbitrary Lagrangian-Eulerian formulation are solved using the finite element method. We vary both the Reynolds number and the Womersley number in order to quantify their effects on the vortex shedding. From the computational results, we calculate the Strouhal number and use this as a metric for the shedding behavior. At high Reynolds numbers, the flow is characterized by the presence of distinct vortex roll cells with long persistence times. At high Womersley numbers, these roll cells are increasingly likely to interact with the cylinder as it reverses direction and translates back across the channel.

8:26AM G12.00003 A Lagrangian Coherent Structures Analysis of the Unsteady Wake Behind a Circular Cylinder , MATTHEW ROCKWOOD, Syracuse University, JACOB MORRIDA[1], Notre Dame, MELISSA GREEN, Syracuse University — The experimentally measured unsteady wake behind a circular cylinder was studied and compared with numerical results. The location and evolution of coherent structures, or vortices, in the flow were analyzed to facilitate the understanding of the vortex shedding physics in the near wake region. This understanding is critical to the control of vortex shedding from bluff bodies. The two-component velocity data was collected using a DPIV measurement system, and Eulerian vortex criteria were applied along with a Lagrangian coherent structures (LCS) analysis to determine the properties of the wake. The LCS analysis utilizes the Finite Time Lyapunov Exponent (FTLE) method to objectively determine the locations of vortex boundaries in the flow. This technique offers new insight on the development of the unsteady wake, and shows an objective change in curvature of the LCS in the region where the new vortex will form before traditional Eulerian techniques show any changes. This information can be used to highlight regions to be targeted by flow control techniques.

[1]Work was completed as a student at Syracuse University.

8:39AM G12.00004 Investigating wake topology of a single step cylinder with tomographic PIV , SERHIY YARUSEVYCH, University of Waterloo, SINA RAFATI, FULVIO SCARANO, TU Delft — Wake vortex shedding from a single step cylinder is investigated experimentally using Tomographic Particle Image Velocimetry (TOMO PIV). The model geometry is comprised of two circular cylinders of different diameters joined concentrically. Experiments are conducted in a low-speed wind tunnel for a range of cylinder diameter ratios $1.14 \leq D/d \leq 2.67$ and Reynolds numbers $2000 \leq Re_D \leq 5000$. The employed TOMO PIV system consists of six CCD cameras subtending an arc and an Nd:YAG laser. LaVision DaVis 8 is used for image acquisition and processing. For the range of parameters investigated, turbulent vortex shedding occurs in the single-step cylinder wake. The difference in diameters leads to a variation in vortex shedding frequency, producing complex three-dimensional vortex interactions in the wake region downstream of the step. The use of TOMO PIV enables quantitative visualization and analysis of the attendant intricate vortex dynamics. Vortex filaments are visualized by the Q-criterion, and the topology of recurring vortex patterns is investigated. Reduced order modeling is used to identify dominant vortex interactions, providing added insight into the wake development. The results are used to reconstruct salient topological features of the near wake region and to investigate the effect of diameter ratio and Reynolds number on the wake topology.

8:52AM G12.00005 The turbulent wake of a submarine model at varying pitch and yaw angle , ANAND ASHOK, Princeton University, TYLER VAN BUREN, Rensselaer Polytechnic Institute, ALEXANDER SMITS, Princeton University and Monash University — Experiments are reported to examine the effects of pitch and yaw angle on the mean flow and turbulence in the wake of an axisymmetric DARPA SUBOFF submarine model. Measurements in the wake were performed at a Reynolds number based on the length of 2.4×10^6. Three component velocity measurements were taken at eight cross-stream planes, downstream of the trailing edge of the model ($2 < x/D < 26$), using Stereoscopic Particle Image Velocimetry. The pitch and yaw angles were in the range 0 to $\pm$ 10°. Two-point, crossed wire measurements in the wake of the same submarine model in the axisymmetric configuration over a wide range ofnumbers based on the length between 1×10^6 and 67×10^6 are also presented. Work supported by ONR Grant N00014-13-1-0174.

9:05AM G12.00006 Shedding characteristics along the span in the wake of a low-aspect-ratio pyramid , ZAHRA HOSSEINI, MOUHAMMAD EL HASSAN, ROBERT MARTINUZZI, University of Calgary — The aim of the present work is to extract the 3D vortical structures in the wake of a low-aspect-ratio pyramid to study the cellular shedding patterns and interactions between structures with different scales along the span. The velocity fields measured, using Time Resolved PIV, at 2D planes in the wake shows the formation of anti-symmetric Karman shedding and instances of symmetrically positioned structures that interrupt the shedding. In such instances the Reynolds stresses drop significantly. The coherent vortical structures were extracted using the first three POD modes, the first two composing a harmonic pair and the third a symmetric mode capturing the low frequency modulations. The energy of the third mode is almost constant along the span which can be related to the spanwise coupling of the structures. The energy of the harmonic pair is significantly larger at the base but drops quickly at higher spans and becomes comparable to that of the third mode. In the symmetric configurations, the harmonic pair amplitude drops sharply and the symmetric mode amplitude is rather high. To understand the mechanisms that result in such interruptions, the 3D structures will be reconstructed from isolated planar measurements using a low order model based on the most dominant POD modes and their correlation with the surface pressure.

9:18AM G12.00007 Vortices Behind Asymmetric Bodies Forming Closed Wakes , ALAN ELCRAT, KEN MILLER, Wichita State University, LUCA ZANNETTI, Politecnico di Torino — We describe flows past bodies in 2D inviscid flow which are uniform at infinity, and in which there are two point vortices standing in equilibrium with the flow. The bodies are not symmetric, and in general there is a circulation around the body plus vortices. This model then has three degrees of freedom which can be reduced by specifying separation points on the body. In particular for bodies with sharp corners the Kutta condition can be imposed at these corners. There results a model for the wake in which, in general, there is flow from upstream infinity to downstream infinity through the recirculation region ie the separating streamlines do not rejoin. The wake is not closed. We give examples in which the remaining free parameters can be used to force closure of the wake ie to have a closed recirculation region. These include arcs for which the Kutta condition is imposed at each end and Joukouski airfoils with the Kutta condition imposed at the trailing edge.

9:31AM G12.00008 Circulation shedding in viscous starting flow past a flat plate , MONIKA NITSCHE, University of New Mexico, LING XU, Georgia State University — Numerical simulations of viscous flow past a flat plate moving in direction normal to itself reveal details of the vortical structure of the flow. At early times, most of the vorticity is attached to the plate. We introduce a definition of the shed circulation at all times and show that it indeed represents vorticity that separates and remains separated from the plate. Scaling laws for the shed circulation, and contributions to the shedding rate across various boundary components are presented, as well as their dependence on Reynolds number. The simulations provide benchmark results to evaluate simpler separation models such as point vortex and vortex sheet models. A comparison with vortex sheet results is included.

9:44AM G12.00009 Pressure-gradient Mechanism for Vortex Shedding in External Flows[1] , MICHAEL BOGHOSIAN, KEVIN CASSEL, Illinois Institute of Technology — In our previous research, a pressure-gradient mechanism is identified as a likely cause of vortex splitting and shedding in constricted two-dimensional channel flows. We now find this mechanism present in the canonical two-dimensional external flows of (a) the cylinder in crossflow and (b) the flow over a step via numerical simulations of the unsteady, two-dimensional, incompressible Navier–Stokes equations. The details of the pressure-gradient mechanism are presented for Reynolds numbers typically found in the literature.

[1]NIH grant R01DK090769

9:57AM G12.00010 The Interplay of Acceleration and Vorticity Fields in the Tip Region of Massively-Separated Flows , DAVID RIVAL, JOCHEN KRIEGSEIS, University of Calgary — The influence of seemingly analogous plate kinematics (plunge vs. tow) on instantaneous forces has been investigated. Simultaneous measurements by means of three-dimensional particle tracking velocimetry (3D-PTV) and a six-component force/moment sensor have been performed. Despite identical effective shear-layer velocities and effective angles of attack, the force histories vary between the two cases (plunge and tow). To uncover this discrepancy, a combined analysis of vorticity, Lagrangian (total) fluid acceleration and vortex-force contribution (Lamb vector) has been performed. It is found that leading-edge vortex (LEV) and tip vortex (TV) formation are nearly identical during the acceleration phase for both cases. However, at the end of acceleration the tow LEV rolls off the plate. As such, the development of vortex force also ceases once this roll-off process begins. Also TV strength as well as its relative positioning to the plate surface influences the instantaneous force. Based on a Lamb-vector analysis of the TV, the present work provides insight into the underlying cause-effect relation. Particularly, it is demonstrated that the sensitivity of the resulting vortex-force formation is dependent on the interplay between streamwise vorticity and spanwise (inboard) velocity.

Monday, November 25, 2013 8:00AM - 9:57AM –

Session G13 Vortex Dynamics and Vortex Flows V 301 - Amy Lang, University of Alabama

8:00AM G13.00001 Unsteadiness of Flow Structure on Low Swept Delta Wing[1] , MOHAMMADREZA ZHARFA, ILHAN OZTURK, MEHMET METIN YAVUZ, Middle East Technical University — The flow structure of low-sweep 35 degree delta wing has been analyzed experimentally using flow visualization and flow measurement techniques. Laser illuminated smoke visualization, Laser Doppler Anemometry (LDA), and surface pressure measurements are performed to understand the steady and unsteady behavior of the flow regimes. Reynolds number varying from 10,000 to 100,000 and attack angles varying from 3 to 10 are tested. For the corresponding Reynolds Numbers and attack angles, prestall and poststall regimes are identified. The amplitude and frequency of the pressure and velocity fluctuations at different locations are compared with the regions of reattachment, vortex breakdown, and stall. Using statistics and spectral analysis, the unsteadiness of flow is studied in detail. Both the lift performance of the wing and the regions possibly exposed to surface buffeting are tried to be interpreted. Using the results of the study, an effective active flow control strategy to delay stall and to reduce surface buffeting is going to be determined next.

[1]The project was supported by the Turkish Scientific and Technological Research Council of Turkey (3501 - 111M732).

8:13AM G13.00002 Flow Structure over Moderate Swept Delta Wing: *Effects of Reynolds Number and Attack Angle* , ILHAN OZTURK, MOHAMMADREZA ZHARFA, MEHMET METIN YAVUZ, Middle East Technical University — Recent investigations have revealed the appearance of a distinctive type of leading edge vortex, dual vortex structure, over simple delta wing planforms having moderate sweep angles. Flow over a moderate swept 45-degree wing has been investigated using laser illuminated smoke visualization, Laser Doppler Anemometry (LDA), and surface pressure measurements. The effects of Reynolds number and attack angles on dual vortex structure, vortex breakdown, and poststall regime are reported. The footprint of flow regimes on the surface of the planform is captured by the pressure measurements, and the lift performance of the wing is tried to be extracted. The relation between surface pressure fluctuations and near surface velocity fluctuations is investigated. The reattachment region of the separated shear layer on the surface, vortex breakdown, and stall regime are studied with considering the aforementioned relation, which will enlighten some of the aspects of the buffeting on the wing planform.

8:26AM G13.00003 Vorticity Transport in the Leading Edge Vortex of a Plunging Airfoil[1] , AZAR ESLAM PANAH, JAMES AKKALA, JAMES BUCHHOLZ, University of Iowa — The development of the leading edge vortex on a plunging flat plate airfoil is investigated using time-resolved particle image velocimetry and time-resolved surface pressure measurements. Interaction of the leading-edge vortex with the surface of the plate results in the creation of a secondary vortex similar to that found in many other flows such as those over delta wings. Temporal and spatial variations in the surface vorticity flux are computed from the time-varying pressure distributions on the surface of the plate. The net circulation resulting from this boundary flux is smaller in magnitude than that from the leading edge shear layer, but of the same order of magnitude. Entrainment of secondary vorticity into the leading-edge vortex results in cross-cancelation within the leading-edge vortex, weakening the leading edge vortex.

[1]The authors gratefully acknowledge support from AFOSR under grant number FA9550-11-1-0019.

8:39AM G13.00004 ABSTRACT WITHDRAWN –

8:52AM G13.00005 Three Dimensional Vortex Wake Structure of Flapping Wings in Hovering Flight , BO CHENG, YUN LIU, XINYAN DENG, School of Mechanical Engineering, Purdue University, BIO-ROBOTICS LAB TEAM — Flapping wings create complex vortex structures in the wake, as the vortices of one wing stroke shed periodically and travel downwards with the induced flow. However, the detailed three-dimensional vorticity distribution and evolution in the far wake are scarcely understood experimentally. In this study, the three-dimensional vortex wake structure in both the near and far field of a dynamically-scaled flapping wing was investigated experimentally, using volumetric three component velocimetry. Summarily, the overall result of the wing action is to create a coherent vortex structure consisting of a tip vortex (TV), trailing-edge shear layer (TESL) and leading-edge vortex (LEV). The shed TESL rolls up into a root vortex (RV); together with the TV in the wake, they contracts radially but stretch tangentially in the wake. Concurrently, the downwash is distributed in an arc-shaped region enclosed by the stretched tangential vorticity of TVs and RVs. Overall, a closed vortex ring structure is not observed in the current study, because there is no well-established starting and stopping vortex structures that smoothly connect to TV and RV. Finally, evaluation of the vorticity transport equation shows that both TV and RV, while convected downwards, undergo vortex stretching, a three-dimensional phenomenon in rotating flows. It also confirms that the vorticity evolution is dominated by convection with secondary tilting/stretch effects, while the magnitude of vorticity dissipation is negligible.

9:05AM G13.00006 Volumetric visualization of the near and far field wake in flapping wings[1] , YUN LIU, BO CHENG, XINYAN DENG, School of Mechanical Engineering, Purdue Unversity, BIO-ROBOTICS LAB TEAM — The flapping wings of flying animals create complex vortex wake structure, understanding its spatial and temporal distribution is fundamental to animal flight theory. In this study, we applied the volumetric 3-component velocimetry to capture both the near- and far-field flow generated by a pair of mechanical flapping wings. For the first time, the complete three-dimensional wake structure and its evolution throughout a wing stroke were quantified and presented. The general vortex wake structure maintains a quite consistent form: vortex rings in the near-field and two shear layers in the far-field. In specific, vortex rings shed periodically from the wings and are linked to each other in successive strokes. In the far-field, the shed vortex rings evolve into two parallel shear layers with dominant vorticity convected from tip and root vortices. The shear layers are nearly stationary in space compared to the periodic vortex rings shed in the near field. In addition, downwash passes through the centers of the vortex rings and extends downward between the two shear layers.

[1]This work is supported by AFOSR.

9:18AM G13.00007 Influence of wing tip morphology on vortex dynamics of flapping flight , SWATHI KRISHNA, KAREN MULLENERS, Leibniz Universitaet Hannover — The mechanism of flapping wing flight provides insects with extraordinary flight capabilities. The uniquely shaped wing tips give insects an edge in flight performance and the interaction between the leading edge vortices and wing tip vortices enhance their propelling efficiencies and manoeuvrability. These are qualities that are sought after in current-day Micro Air Vehicles. A detailed understanding of the vortex dynamics of flapping flight and the influence of the wing tip planform is imperative for technical application. An experimental study is conducted to investigate the effects of different wing tip planforms on the formation, evolution and interaction of vortical structures. We thereby focus on the interaction between the coherent structures evolving from the leading edge and the wing tip during pitching and flapping motions.The spatial and temporal evolution of the three-dimensional flow structures are determined using Scanning (Stereo) Particle Image Velocimetry and an in-depth coherent structure analysis. By comparing the vortex dynamics, the aerodynamic performance of various wing tip planforms are evaluated.

9:31AM G13.00008 Vortex shedding in flow past an airfoil using boundary layer approximation , XINJUN GUO, SHREYAS MANDRE, Brown University — We present an extension of the Kutta condition using matched asymptotic expansion applied to the Navier-Stokes equation. The goal is to study the influence of unsteady fluid dynamical effects like leading edge vortex, unsteady boundary layer separation, etc. in flow around a solid body. In order to describe accurately the location and strength of vortex shedding, we solve the simplified Navier-Stokes equations in the form of boundary layer approximation in the thin inner region close to the solid body. In the outer region far from the structure, the vortex methods are applied, which significantly reduces the computational cost compared to CFD in the whole domain.

9:44AM G13.00009 Performance of Piezoelectric Energy Harvesters in Isotropic Turbulence[1] , AMIR DANESH-YAZDI, OLEG GOUSHCHA, NIELL ELVIN, YIANNIS ANDREOPOULOS, The City College of the City University of New York — A piezoelectric harvester beam is used to extract energy from the turbulence in a surrounding fluid. An experimental investigation is carried out in a large scale wind tunnel in which turbulence-generating grids of varying dimensions are used to excite a flexible cantilever beam with a piezoelectric patch. The beam is instrumented with a strain gauge and the strain and voltage generated by the piezoelectric material are recorded as a function of time at various distances from the grid. We observe that the presence of the beam breaks the flow isotropy in the near field but the pressure forcing content retains some of the isotropy features. The pertinent parameters that influence the voltage output and performance of the beam are identified as (1) the dimensionless distance of the beam from the grid with respect to the grid size and (2) the dimensionless length of the beam. The voltage output is also shown to obey an exponential decay law with respect to the dimensionless distance parameter. A theoretical solution to the voltage output and tip displacement is also suggested and the results are compared with experimental data.

[1]Sponsored by NSF Grant CBET #1033117.

Monday, November 25, 2013 8:00AM - 10:10AM –

Session G14 Experimental Techniques IV: PIV - Uncertainty/Microscopic 302 - Pavlos Vlachos, Virginia Polytechnic Institute and State University

8:00AM G14.00001 Uncertainty estimation for Stereo-Particle Image Velocimetry measurements , SAYANTAN BHATTACHARYA, Purdue University, BRETT MEYERS, MATTHEW GIARRA, RODERICK LA FOY, Virginia Polytechnic Institute and State University, PAVLOS VLACJOS, Purdue University — Stereographic Particle Image Velocimetry (SPIV) is a standard method of estimating three-component fluid velocity fields from a two-dimensional field of view using two viewing angles. SPIV techniques involve a series of procedures such as camera calibration, image de-warping, velocity field reconstruction, etc., and the contribution of each step to the overall uncertainty of the measurement is not well understood. Previous efforts have been made to quantify errors involved at each stage of the 3D velocity reconstruction, but have fallen short of a rigorous analysis of the combination of errors for a range of parameters. Such analysis is performed herein. In the present work the Type A uncertainty is evaluated for each step of an SPIV method (involving self calibration and both 3D calibration-based reconstruction and geometric reconstruction) for a set of simulated images. A simulated vortex ring image set was used as a test case and the particle seeding density, light sheet thickness, image magnification, and viewing angles were varied parametrically. Propagation of systematic and random standard uncertainty using both Taylor series and Monte-Carlo method was performed. The results are also compared with prior 2D PIV uncertainty analysis.

8:13AM G14.00002 Evaluation of multi-pulse PIV for spatial resolution, velocity accuracy and acceleration measurement , LIUYANG DING, RONALD ADRIAN, Arizona State University, SIVARAM GOGINENI, Spectral Energies LLC, KATHY PRESTRIDGE, Los Alamos National Laboratory — The performance of multi-pulse PIV is numerically evaluated based on PTV simulation. We compare triple-pulse and quadruple-pulse to conventional double-pulse PIV regarding their performance on spatial resolution, velocity and acceleration measurement. The optimization is achieved to minimize the combined error in position, velocity and acceleration. Multi-pulse technology is then tested by measuring simultaneous velocity and acceleration fields of a round impinging air jet. Experimental results from triple-pulse and quadruple-pulse are compared and discussed in terms of the accuracy and performance.

8:26AM G14.00003 3 Component PIV Uncertainty[1] , SCOTT WARNER, BARTON SMITH, Utah State University — The random uncertainty of 2-component (2C) Particle Image Velocimetry (PIV) has recently been addressed in three unique methods called the Uncertainty Surface Method (USM) from Utah State University, Image Matching (IM) method from Lavision and Delft, and correlation Signal to Noise Ration (SNR) methods from Virginia Tech. Since 3C (stereo) Particle Image Velocimetry (PIV) velocity fields are derived from two, 2C fields, random uncertainties from the 2C fields clearly propagate into the 3C field. In this work, we will demonstrate such a propagation using commercial PIV software and the USM method, although the propagation works similarly for any 2C random uncertainty method. Stereo calibration information is needed to perform this propagation. As a starting point, a pair of 2C uncertainty fields will be combined in exactly the same manner as velocity fields to form a 3C uncertainty field using commercial software. Correlated uncertainties between the components in the two 2C fields will be addressed. These results will then by compared to a more rigorous propagation, which requires access to the calibration information.

[1]Thanks to the Nuclear Science & Technology Directorate at Idaho National Laboratory. The work was supported through the U.S. Department of Energy, Laboratory Directed Research & Development grant under DOE Contract 122440 (Project Number: 12-045).

8:39AM G14.00004 Effects of Spatial Alignment in Stereo Particle Image Velocimetry , BARTON SMITH, Utah State University, STEVEN BERESH, Sandia National Laboratory — We seek to quantity errors in stereo Particle Image Velocimetry (PIV) as a function of laser sheet thickness and camera angle. Simultaneous stereo PIV measurements of a simple free jet were obtained from narrow and wide camera angles while a fifth camera viewed the laser sheet from 90 degrees to determine the two-component velocity field free of errors resulting from stereo calibration. Errors in mean velocities were small, but artificially reduced turbulent stresses were generated when self-calibration was not used, owing to a smearing effect that occurs when the two cameras are inadequately registered to each other. This difficulty worsens with increased laser sheet thickness. Spatial error in the calibration process can artificially displace vector fields from the expected origin. Although this typically is small with respect to statistical properties of a data set, it can be prominent when instantaneous snapshots of the velocity field are examined, particularly where the velocity gradient is momentarily large. Furthermore, small scale structures present in the jet flow are distorted by the various PIV systems in a manner that depends on the sheet thickness and camera orientation.

8:52AM G14.00005 Correlation plane statistical analysis for estimation of measurement uncertainty for Particle Image Velocimetry , ZHENYU XUE, Department of Mechanical Engineering, Virginia Tech, JOHN CHARONKO, Los Alamos National Laboratory, PAVLOS VLACHOS, Department of Mechanical Engineering, Virginia Tech — Early development of Particle Image Velocimetry (PIV) methods did not involve quantification of measurement uncertainty, which in result created skepticism about the reliability of PIV. Quantification of PIV uncertainty is complex because coupled sources are involved in PIV measurement. Recently several attempts have been proposed. However, most of those methods were "posteriori" methods: deducing the uncertainty from post-processing of recorded images, or using observed relationships between metrics calculated from images, flow field and the resulting error distribution. Here we propose a novel theoretical and statistical PIV uncertainty estimation approach. It is based on the notion that the correlation plane represents the probability distribution function (PDF) of all possible particle displacements convoluted with particle shape information. The PDF can be obtained by de-convolving the particle information from original correlation plane. Knowing the primary peak of correlation plane indicates the most probable displacement, and the PDF, standard deviation of measured displacement, i.e. the uncertainty, can be calculated by computing the second order moment about the most probable displacement. We will present theoretical and statistical foundations of this method, we will validate each performance with synthetic image sets, and finally we will show its application on real experiment data.

9:05AM G14.00006 Nano-scale velocimetry with Bessel Beam Microscopy , CRAIG SNOEYINK, Texas Tech University — Bessel Beam Microscopy is a unique imaging technique that places an axicon in the imaging path of a microscope. Here we will discuss recent advances in this technique including single-acquisition with 40% improved spatial resolution and single-view three-dimensional particle tracking with greatly enhanced depth resolution. These capabilities lead to enhanced resolution in velocimetry techniques. For example, when using BBM to perform Particle Image Velocimetry (PIV) the increased image spatial resolution allows for a corresponding increase in velocity field spatial resolution. The greatly increased depth resolution, on the order of 100 nm with a 10x objective, can greatly increase the spatial resolution of Particle Tracking Velocimetry (PTV) measurements.

9:18AM G14.00007 Development of real time digital holographic microscope for cell flow interactions using a High Performance Computing (HPC) cluster , AVESTA HOJJATI, MEHDI MOLAEI, JIAN SHENG, Texas Tech University — Real-time imaging and analysis of 3D cell migration and locomotion is crucial to understand the underlying physics of cell environment interactions. In addition, such a microscopy would provide vital diagnostic capability in cell detection, particle sorting and drug screening with large throughput. However, 3D holographic imaging and subsequent analysis are computational intensive and up-to-date prohibitive for real-time applications. With the advances in high performance computing, we are developing a real-time digital holographic microscope (DHM) that includes an in-line DHM, a large format CCD camera, and a 24-node windows-based HPC cluster. The cluster is organized as the master-slave parallel computing paradigm with Message Passing Interface (MPI) as its communication protocol. The holograms are recorded, streamed and analyzed by the HPC cluster in real time, the 3D distributions and in focus images are rendered back on the data acquisition computer. The system will be applied to study marine protest interacting with oil droplets. Supports from GoMRI are acknowledged.

9:31AM G14.00008 Real and virtual image separation in digital in-line holography microscopy by recording two parallel holograms[1] , HANGJIAN LING, JOSEPH KATZ, Johns Hopkins University — Maintaining high magnification and micron resolution in applications of digital in-line holography microscopy for 3D velocity measurements requires a hologram plane located very close or even within the sample volume. Separation between overlapping real and virtual images becomes a challenge in such cases. Here, we introduced a simple method based on recording two holograms through the same microscope objective that are separated by a short distance from each other. When the same particle fields are reconstructed from the two holograms, the real images overlap, whereas virtual images are separated by twice the distance between hologram planes. Thus, real and virtual images can be easily distinguished. Due to the elongation of the reconstructed particle in the axial direction, the distance between hologram planes is selected to exceed the elongated traces. This technique has been applied to record 3D traces of thousands of 2 um particles in a $0.5 \times 0.5 \times 0.5$ mm sample volume using hologram planes separated by 27 um. Experimental setup, alignment and data analysis procedures, including reconstruction, calibration, particles segmentation and precision particles positioning will be discussed.

[1]Sponsored by ONR

9:44AM G14.00009 Turbulent Boundary Layer Facility to Investigate Superhydrophobic Drag Reduction[1] , JAMES W. GOSE, MARC PERLIN, STEVEN L. CECCIO, University of Michigan — Recent developments in superhydrophobic surfaces have led to potential economic and environmental benefits, perhaps most notably in skin-friction drag reduction. A team from the University of Michigan has developed a recirculating turbulent boundary layer facility to investigate the reduction of drag along engineered superhydrophobic surfaces (SHS). The facility can accommodate both small and large SHS samples in a test section 7 mm (depth) x 100 mm (span) x 1000 mm (length). Coupled with an 11.2 kilowatt pump and a 30:1 contraction the facility is capable of producing an average flow velocity of 25 m/s, yielding a Reynolds number of 84,000. Flexure-mounted test samples subjected to shear deflect to a max of 50 microns; movements are measured using a digital microscope composed of a high-resolution camera and a water immersion objective. The setup yields an optical resolution of about one micron whereas sub-micron resolution is achieved by implementing an FFT of two Ronchi rulings. Additional drag measurement methods include pressure drop across the test specimen and PIV measured boundary layers. Additional SHS investigations include the implementation of active gas replenishment, providing an opportunity to replace gas-pockets that would otherwise be disrupted in traditional passive SHS surfaces due to high shear stress and turbulent pressure fluctuations.

[1]The authors recognize the support of ONR.

9:57AM G14.00010 Echo Particle Image Velocimetry Measurements of Liquified Biomass[1] , NICHOLAS DEMARCHI, CHRISTOPHER WHITE, University of New Hampshire — Echo particle image velocimetry (EPIV) is used to acquire planar fields of velocity in pipe flow of liquefied biomass. The biomass studied is pre-treated (i.e., acid washed) corn stover and it is liquefied by enzymatic hydrolysis. The liquefaction process is carried out for various biomass mass loadings (1.5%-15%). For each biomass loading, the fluid's microstructure and rheology are studied and EPIV measurements are acquired. The aim is to demonstrate the usefulness of EPIV to acquire planar fields of velocity in optically opaque flows and to evaluate the effect of particle size, distribution, and mass loading of the dispersed solid phase on the EPIV measurements.

[1]NSF-CBET0846359

Monday, November 25, 2013 8:00AM - 10:10AM –

Session G16 Biofluids: Physiological V - Respiratory System Flows 304 - Brent A. Craven, Pennsylvania State University

8:00AM G16.00001 On locating the obstruction in the human upper airway[1] , YONG WANG, S. ELGHOBASHI, University of California, Irvine — The fluid dynamical properties of the air flow in the human upper airway (UA) are not fully understood at present due to the three-dimensional, patient-specific complex geometry of the airway, flow transition from laminar to turbulent and flow-structure interaction during the breathing cycle. One of the major challenges to surgeons is determining the location of the UA obstruction before performing corrective surgeries. It is quite difficult at present to experimentally measure the instantaneous velocity and pressure at specific points in the human airway. On the other hand, direct numerical simulation (DNS) can predict all the flow properties and resolve all its relevant length- and time-scales. We developed a DNS solver with lattice Boltzmann method (LBM), and used it to investigate the flow in two patient-specific UAs reconstructed from CT scan data. Inspiration and expiration flows through these two airways are studied and compared. Pressure gradient-time signals at different locations in the UAs are used to determine the location of the obstruction.

[1]This work was supported by the National Heart, Lung, and Blood Institute (NHLBI) of the National Institutes of Health (NIH).

8:13AM G16.00002 A fully resolved fluid-structure-muscle-activation model for esophageal transport[1] , WENJUN KOU, Theoretical and Applied Mechanics Program, Northwestern University, AMNEET P.S. BHALLA, Department of Mechanical Engineering, Northwestern University, BOYCE E. GRIFFITH, Leon H. Charney Division of Cardiology, Department of Medicine, New York University School of Medicine, MARK JOHNSON, Department of Biomedical Engineering, Northwestern University, NEELESH A. PATANKAR, Department of Mechanical Engineering, Northwestern University — Esophageal transport is a mechanical and physiological process that transfers the ingested food bolus from the pharynx to the stomach through a multi-layered esophageal tube. The process involves interactions between the bolus, esophageal wall composed of mucosal, circular muscle (CM) and longitudinal muscle (LM) layers, and neurally coordinated muscle activation including CM contraction and LM shortening. In this work, we present a 3D fully-resolved model of esophageal transport based on the immersed boundary method. The model describes the bolus as a Newtonian fluid, the esophageal wall as a multi-layered elastic tube represented by springs and beams, and the muscle activation as a traveling wave of sequential actuation/relaxation of muscle fibers, represented by springs with dynamic rest lengths. Results on intraluminal pressure profile and bolus shape will be shown, which are qualitatively consistent with experimental observations. Effects of activating CM contraction only, LM shortening only or both, for the bolus transport, are studied. A comparison among them can help to identify the role of each type of muscle activation.

[1]The support of grant R01 DK56033 and R01 DK079902 from NIH is gratefully acknowledged.

8:26AM G16.00003 Effect of Time-dependent Pressure Boundary Condition on Flow Transport in a Patient Specific Lung Model during Invasive High Frequency Oscillatory Ventilation , MOHAMMED ALZAHRANY, ARINDAM BANERJEE, Lehigh University — Large eddy simulation was used to investigate gas transport in a human lung (image-based) model during high frequency oscillatory ventilation (HFOV). A time-dependent pressure boundary condition as a function of the flow rate and coupled resistance-compliance was imposed at the outlets. The study was conducted for three different HFOV frequencies of 6, 10 and 15 Hz; a constant tidal volume of 50 ml and various compliance ratios (1, 4 and 10). The results are compared to computations that use traditional boundary conditions (such as pre-specified flow and constant pressure), experimental and gamma scintgraphy results. While traditional pre-specified mass fraction boundary condition failed to capture the Pendelluft flow at regional lung units that are observed in experiments, our modified resistance-compliance based pressure boundary condition was successful in predicting this feature. The impact of compliance ratio and frequency on phase-delay at different lung sections and its effect on secondary flow and turbulence will also be presented.

8:39AM G16.00004 Effect of morphological variability on particle deposition in idealized human airways[1] , ELEANOR LIN, JORGE A. BERNATE, Stanford University, DANIEL A. PARADA SAN MARTIN, Pierre and Marie Curie University, YUZO MAKITANI, California Polytechnic State University, ERIC S. G. SHAQFEH, GIANLUCA IACCARINO, Stanford University — This study is focused on the effects of variability in airway morphology on particle deposition in the lungs, which in turn impacts disease inception and drug delivery. We generated a parameterized geometry of the human airway derived from Lola: a realistic geometry obtained from CT scans (Zhang et. al J AEROSOL SCI 46, 34 (2012)). The upper airway geometry is parameterized using an elliptic model from Xi and Longest (ANN BIOMED ENG 35,560 (2007)), with the glottis modified to a realistic triangular shape, based on measurements taken from Lola. The trachea and bronchi are generated using rules adapted from Kitaoka et. al. (J Appl Physiol 87, 2207-2217 (1999)), with the first 3 generations closely matching those of Lola. We perform simulations corresponding to a full breathing cycle and illustrate the preferential deposition in each generation. In addition, we compared the deposition features in the idealized geometry to those from simulations in the original scanned airways. Perturbations are then applied to the parameterized geometry to study the effects of morphological variability on deposition patterns.

[1]This work is funded by the Army AHPCRC at Stanford.

8:52AM G16.00005 Renal hemodynamics: the influence of the renal artery ostium flow diverter , JENN STROUD ROSSMANN, SCOTT ALBERT, Lafayette College, ROBERT BALABAN, National Institutes of Health — The recently identified renal artery ostium flow diverter may preferentially direct blood flow to the renal arteries, and may also influence flow patterns and recirculation known to be involved in atherogenesis. Three-dimensional computational fluid dynamics (CFD) simulations of steady and pulsatile blood flow are performed to investigate the influence of diverter size and position, and vascular geometry, on the flow patterns and fluid mechanical forces in the neighborhood of the diverter. CFD results show that the flow diverter does affect the blood distribution: depending on the diverter's position, the flow to the renal arteries may be increased or reduced. The results of simulations also show the diverter's effect on the Wall Shear Stress (WSS) distribution, and suggest that the diverter contributes to an atherogenic environment in the abdominal aorta, while being atheroprotective in the renal arteries themselves. These results support previous clinical findings, and suggest directions for further clinical study. The results of this work have direct implications in understanding the physiological significance of the diverter, and its potential role in the pathophysiological development of atherosclerosis.

9:05AM G16.00006 Simulation of the flow field and particle deposition in a realistic geometry of the human airways[1] , JORGE A. BERNATE, ELEANOR LIN, ERIC S.G. SHAQFEH, GIANLUCA IACCARINO, Stanford University — Using the dynamic Smagorinsky sub-grid scale model, we carry out Large Eddie Simulations (LES) of the flow field in a realistic geometry reconstructed from a CT scan of an adult male human subject (Zhang et. al J AEROSOL SCI 46, 34 (2012)). The geometry comprises the oral cavity, larynx, trachea, and bronchi extending to generations 6 to 9. The computed time-averaged flow field is validated with magnetic resonance velocimetry (MRV) measurements obtained in a 3D printed model of the realistic geometry (Andrew J. Banko, Filippo Coletti, Daniele Schiavazzi, Christopher J. Elkins, John K. Eaton, submitted to this conference). The comparison is done at a constant inspiratory flow rate of 60 L/min, at which turbulence is expected to develop. Probing the mean flow, we compare integral factors quantifying the ventilation, the shape of stream-wise velocity profile, and the strength of secondary flows in different branches. Via simulations, we also characterize the unsteadiness of the flow, focusing on the dynamics of the laryngeal jet and its effect on the structure of the flow field and particle deposition patterns.

[1]This work is funded by the Army AHPCRC at Stanford

9:18AM G16.00007 An image-based automatic mesh generation and numerical simulation for a population-based analysis of aerosol delivery in the human lungs[1] , SHINJIRO MIYAWAKI, The University of Iowa, MERRYN H. TAWHAI, The University of Auckland, ERIC A. HOFFMAN, CHING-LONG LIN, The University of Iowa — The authors propose a method to automatically generate three-dimensional subject-specific airway geometries and meshes for computational fluid dynamics (CFD) studies of aerosol delivery in the human lungs. The proposed method automatically expands computed tomography (CT)-based airway skeleton to generate the centerline (CL)-based model, and then fits it to the CT-segmented geometry to generate the hybrid CL-CT-based model. To produce a turbulent laryngeal jet known to affect aerosol transport, we developed a physiologically-consistent laryngeal model that can be attached to the trachea of the above models. We used Gmsh to automatically generate the mesh for the above models. To assess the quality of the models, we compared the regional aerosol distributions in a human lung predicted by the hybrid model and the manually generated CT-based model. The aerosol distribution predicted by the hybrid model was consistent with the prediction by the CT-based model. We applied the hybrid model to 8 healthy and 16 severe asthmatic subjects, and average geometric error was 3.8% of the branch radius. The proposed method can be potentially applied to the branch-by-branch analyses of a large population of healthy and diseased lungs.

[1]NIH Grants R01-HL-094315 and S10-RR-022421, CT data provided by SARP, and computer time provided by XSEDE

9:31AM G16.00008 ABSTRACT WITHDRAWN –

9:44AM G16.00009 A Comparative Study of Airflow and Odorant Deposition in the Mammalian Nasal Cavity , JOSEPH RICHTER, CHRISTOPHER RUMPLE, ALLISON RANSLOW, ANDREW QUIGLEY, Penn State University, BENISON PANG, University of California, Los Angeles, THOMAS NEUBERGER, MICHAEL KRANE, Penn State University, BLAIRE VAN VALKENBURGH, University of California, Los Angeles, BRENT CRAVEN, Penn State University — The complex structure of the mammalian nasal cavity provides a tortuous airflow path and a large surface area for respiratory air conditioning, filtering of inspired contaminants, and olfaction. Due to the small and contorted structure of the nasal turbinals, nasal anatomy and function remains poorly understood in most mammals. Here, we utilize high-resolution MRI scans to reconstruct anatomically-accurate models of the mammalian nasal cavity. These data are used to compare the form and function of the mammalian nose. High-fidelity computational fluid dynamics (CFD) simulations of nasal airflow and odorant deposition are presented and used to compare olfactory function across species (primate, rodent, canine, feline, ungulate).

9:57AM G16.00010 Convective-diffusive particle transport in pulmonary acinar models , PHILIPP HOFEMEIER, JOSUE SZNITMAN, Technion - Israel Institute of Technology — Much of our understanding of the transport and deposition of fine inhaled particles (≤ 1 μm) in the deep regions of the lungs results from numerical simulations that revolve around the central assumption that fine aerosols are mainly influenced by local convective airflows. Recently, it has been noted that aerosol transport in the pulmonary acinus relies however on the complex coupling between convective, diffusive processes, as captured by the appropriate dimensionless particle numbers (Sznitman, *J. Biomech.*, 2013). It is anticipated that for particles in the range of $0.5 - 1$ μm, the coupling of intrinsic particle motion with acinar flow fields ultimately governs deposition outcomes. In an effort to address the influence of convective-diffusive mechanisms on aerosol transport and deposition, we present a detailed investigation of fine particle transport in the absence and presence of stochastic Brownian motion. Further, we study systematically the effects of particle properties (e.g., diameter) as well as acinar lung generation on the ensemble statistics of inhaled fine particles and their deposition. Our findings reveal the intimate coupling between local acinar flow structures and intrinsic particle motion leading to complex irreversible aerosol kinematics.

Monday, November 25, 2013 8:00AM - 10:10AM –

Session G17 Biofluids: Locomotion IV - Liquids; Experiments and Numerical Simulations 305 - Keith Moored, Lehigh University

8:00AM G17.00001 Quantitative analysis of fish wake dynamics using volumetric PIV data , LEAH MENDELSON, ALEXANDRA TECHET, MIT — In the study of swimming hydrodynamics, the fluid impulse in the wake is used to quantify the momentum transferred by the fish as it swims. This impulse is typically computed from planar PIV measurements of the wake circulation and geometry by assuming an axisymmetric vortex ring model. However, in many propulsive and maneuvering scenarios, three-dimensional effects are of substantial importance, and wake features are not often an isolated, symmetric vortex ring. Volumetric PIV data provides a complete measure of the vortex geometry and orientation, and circulation can be determined over multiple planar slices through the volume. Using sample datasets obtained from synthetic aperture PIV (SAPIV), we demonstrate how the availability of volumetric PIV data enables more detailed analysis of hydrodynamic impulse and characterize the uncertainty created by planar measurements. Special attention is paid to unsteady maneuvering behaviors that generate asymmetric and linked wake features.

8:13AM G17.00002 On the effect of flexibility on the performance of a bio-inspired fin[1] , STEFANO CHIAZZA, FLORIAN H.J. BREMER, Princeton University, ALEXANDER J. SMITS, Princeton University and Monash University — Experiments are performed to examine the flowfield characteristics of bio-inspired fins of different flexibility. The measurements are performed in a water channel at a fixed frequency of oscillation and different flow velocities covering the free-swimming condition. Thrust and efficiency measurements are complemented by PIV and flow visualizations studies. The wake topology is analyzed at different Strouhal numbers for each flexibility, and the differences between accelerating, decelerating, and free-swimming fins are identified.

[1]This work was supported by the ONR through MURI Grant N00014-08-1-0642 (Program Manager: Dr. Bob Brizzolara).

8:26AM G17.00003 ABSTRACT WITHDRAWN –

8:39AM G17.00004 Scaling the hydrodynamic performance of heaving flexible panels[1] , DANIEL QUINN, Princeton University, GEORGE LAUDER, Harvard University, ALEXANDER SMITS, Princeton University and Monash University — We present an experimental investigation of flexible panels actuated with heave oscillations at their leading edge. Our methods consist of kinematic video analysis, particle image velocimetry (PIV), and direct force measurements. Both the trailing edge amplitude and the mode shapes of the panel are found to scale with dimensionless ratios originating from the Euler-Bernoulli beam equation. Time-averaged thrust increases with heaving frequency, but also shows localized boosts around resonant frequencies where the trailing edge amplitude is maximized. For a constant heave amplitude, the time-averaged thrust coefficient is shown to be a function of Strouhal number over a wide range of conditions. Instantaneous thrust shows two peaks per oscillation cycle, occurring during the mid up- and down-stroke of the leading edge.

[1]Supported by the Office of Naval Research under Program Director Dr. Bob Brizzolara, MURI grant number N00014-08-1-0642, as well as by the National Science Foundation, Grant DBI 1062052

8:52AM G17.00005 Lift enhancement by spanwise oscillation in forward translation of a rectangular wing at low Reynolds number , SHIZHAO WANG, XING ZHANG, GUOWEI HE, Institute of Mechanics, Chinese Academy of Sciences — The effects of bat-inspired spanwise oscillation on the aerodynamic performance of a translating rectangular wing at Reynolds number 300 are investigated numerically. The spanwise oscillation of the wing is in the sinusoidal form, with two control parameters being the amplitude and the frequency. Compared with the wings in pure translation, it is observed that in range of the parametric space onsidered in this work, spanwise oscillation is effective in enhancing lift, augmenting lift-drag ratio. To elucidate the mechanism of lift enhancement, the wake structure behind a wing in the combined motion of translation and spanwise oscillation is compared with that behind a purely translating wing. The phenomenon of lift enhancement in oscillating wings is also analysed by using an approximation formula that associates the lift force with Lamb's vector. It is found that spanwise oscillation produces compact and strong side-edge vortices (SEVs) which apply large downward induced velocity on the LEVs and press them onto the upper surface of the wing.The re-positioning of LEVs (due to the presence of SEVs) benefits lift generation. The novel wing kinematics (combination of translation and spanwise oscillation) of this work has the potential for application in micro air vehicles (MAVs).

9:05AM G17.00006 Hydrodynamics of foils swimming in a side-by-side configuration[1] , PETER DEWEY, Princeton University, KEITH MOORED, Lehigh University, DANIEL QUINN, Princeton University, ALEXANDER SMITS, Princeton University and Monash University — Experimental and computational results are presented on a pair of hydrofoils undergoing pitch oscillations in a side-by-side configuration. The time-averaged forces and propulsive efficiency are independently measured for each foil for a range of separation distances and oscillation phase differentials between the two foils. The results are compared to an isolated foil to determine if the presence of a second foil can yield an improvement to the propulsive characteristics of the system. While the exact performance of the side-by-side foils is strongly dependent on the separation distance and phase differential between the foils, it is found that under certain configurations an enhancement in net thrust is achieved by the presence of a second foil. The wake patterns shed by the foils as they oscillate are also examined and compared to the propulsive characteristics. A series of four stable wake configurations are observed that depend on the phase differential between the foils.

[1]Supported by the Office of Naval Research under Program Director Dr. Bob Brizzolara, MURI grant number N00014-08-1-0642

9:18AM G17.00007 Mimicking fish-like kinematics using fluid-structure interactions , BENJAMIN THIRIA, SOPHIE RAMANANARIVO, RAMIRO GODOY-DIANA, PMMH-ESPCI, PMMH-ESPCI TEAM — We present here a new experiment on a clamped-free elastic slender plate under local harmonic forcing. In air, the solution consists in a sum of standing waves, whose frequencies, wavelengths and mode shapes are given by the Young modulus and the geometry. In more dense fluid, as water, and for specific parameters of the experiment, the solution switch from this standing waves solution to a pure propagating behavior leading to a fish-like kinematics. The existence of this regime allows to simply generate a propagating wave in a finite elastic medium avoiding a complex implementation of synchronized local perturbations all along the body. We show that the triggering of the propagating solution is due to the nonlinear nature of the fluid damping.

9:31AM G17.00008 Dynamically Coupled Fluid-Body Interactions with a Versatile Multi-Domain Immersed Boundary Library[1] , CHENGJIE WANG, JEFF D. ELDREDGE, University of California, Los Angeles — A computational algorithm is developed to simulate dynamically coupled interaction between fluid and rigid bodies. The basic computational framework is built upon a multi-domain immersed boundary method library, whirl, developed in previous work. The multi-domain approach inspired by Colonius and Taira (2008) is intended to cover a large simulation domain with multiple bodies by using a hierarchy of nested domains with different sizes and grid resolutions. This approach enables a versatile and economical use of computational resources. The library hides the details of the fully parallel treatment from the code developer, leading to simple construction of 2D or 3D solvers. In the present case, the incompressible Navier-Stokes equations are solved in vorticity-streamfunction form. The rigid body equations of motion are assembled with the flow equations with immersed boundary forces. The resulting saddle point system is solved in strongly coupled form by the Schur complement reduction method. The resulting scheme is tested by several numerical examples, such as vortex induced oscillation of a cylinder in 2D. Preliminary results are shown for the flapping of a low-aspect-ratio hinged wing at low Reynolds number. The results are compared with previous simulations and experiments.

[1]Supported by AFOSR award number FA9550-11-1-0098.

9:44AM G17.00009 Clear Delineation of Added-Mass and Vortex- Induced Forces Generated by Flapping Wing[1] , CHAO ZHANG, Johns Hopkins University, TYSON HEDRICK, University of North Carolina at Chapel Hill, RAJAT MITTAL, Johns Hopkins University — The force and moment experienced by a body immersed in a fluid depends strongly on its motion (trajectory, acceleration, rotation etc) as well as nearby flow structures such as vortices and boundary layers. A number of past studies have attempted to delineate the relative contribution of various components such as added-mass, attached and shed vortices, and viscous stresses on the total force produced by biological and bioinspired flapping wings. In the current study, we extend a previous analysis (M. S. Howe, 1995) to more precisely delineate the contributions of each of these components to the total force. The analysis is applied via high-fidelity computational fluid dynamics models and we use this analysis to shed light on the various flow mechanisms and features that are responsible for lift generation in insects over a range of scales.

[1]This research is supported by AFOSR.

9:57AM G17.00010 Drag on swimming flexible foils[1] , VERONICA RASPA, SOPHIE RAMANANARIVO, BENJAMIN THIRIA, RAMIRO GODOY-DIANA, PMMH Lab, ESPCI ParisTech, CNRS — We study experimentally the swimming dynamics of thin flexible foils in a self-propelled configuration. Measurements of swimming speed and propulsive force are performed, together with full recordings of the elastic wave kinematics and particle image velocimetry around the swimming foils. We discuss the general problem of drag in undulatory swimming using a bluff-body type model. Our results suggest that a major contribution to the total drag is due to the trailing longitudinal vortices that roll-up on the lateral edges of the foil. Additionally, changing the aspect ratio of the foils allows us to discuss quantitatively the role of the added mass term in Lighthill's elongated-body theory for thrust production in undulatory swimming.

[1]We acknowledge support by EADS Foundation through project "Fluids and elasticity in biomimetic propulsion."

Monday, November 25, 2013 8:00AM - 10:10AM –

Session G18 Biofluids: Locomotion V - Swimming Experiments 306/307 - John Bush, Massachusetts Institute of Technology

8:00AM G18.00001 Flying fish accelerate at 5 G to leap from the water surface , PATRICIA YANG, SULISAY PHONEKEO, KE XU, Georgia Institute of Technology, SHUI-KAI CHANG, National Sun Yat-sen University, DAVID HU, Georgia Institute of Technology — Flying fish can both swim underwater and glide in air. Transitioning from swimming to gliding requires penetration of the air-water interface, or breaking the "surface tension barrier," a formidable task for juvenile flying fish measuring 1 to 5 cm in length. In this experimental investigation, we use high-speed videography to characterize the kinematics of juvenile flying fish as they leap from the water surface. During this process, which lasts 0.05 seconds, flying fish achieve body accelerations of 5 times earth's gravity and gliding speeds of 1.3 m/s, an order of magnitude higher than their steady swimming speed. We rationalize this anomalously high speed on the basis of the hydrodynamic and surface tension forces and torques experienced by the fish. Specifically, leaping fish experience skin friction forces only on the submerged part of their body, permitting them to achieve much higher speeds than in steady underwater swimming. We also perform experiments using a towed flying fish mimc to determine optimality of various parameters in this process, including body angle and start position with respect to the water surface.

8:13AM G18.00002 Flexibility increases lift on passive fluttering wings , DANIEL TAM, TU Delft, JOHN BUSH, MIT — We examine the influence of flexibility on the side-to-side fluttering motion of passive wings settling under the influence of gravity. This effect is examined through an experimental investigation of deformable rectangular wings falling in a water tank. Our results demonstrate the existence of an optimal flexibility, for which flexible wings remain flying twice longer and hence settle twice slower compared to rigid wings of identical mass and geometry. Flow visualizations and measurements provide key insight to elucidate the role of flexibility in generating increased lift and wing circulation by shedding additional vorticity at the turning point. Theoretical scalings are derived from a reduced model of the flight dynamics in qualitative and quantitative agreement with experiments. These scalings rationalize the strong positive correlation between flexibility and time of flight.

8:26AM G18.00003 Synthetic C-start maneuver in fish-like swimming[1] , R. ZENIT, Universidad Nacional Autonoma de Mexico, R. GODOY-DIANA, PMMH UMR7636 CNRS, ESPCI ParisTech, UPMC (Paris 6), U. Paris Diderot (Paris 7) — We investigate the mechanics of the unsteady fish-like swimming maneuver using a simplified experimental model in a water tank. A flexible foil (which emulates the fish body) is impulsively actuated by rotating a cylindrical rod that holds the foil. This rod constitutes the head of the swimmer and is mounted through the shaft of the driving motor on an rail with an air bearing. The foil is initially positioned at a start angle and then rapidly rotated to a final angle, which coincides with the free-moving direction of the rail. As the foil rotates, it pushes the surrounding fluid, it deforms and stores elastic energy which drive the recovery of the straight body shape after the motor actuation has stopped; during the rotation, a trust force is induced which accelerates the array. We measure the resulting escape velocity and acceleration as a function of the beam stiffness, size, initial angle, etc. Some measurements of the velocity field during the escape were obtained using a PIV technique. The measurements agree well with a simple mechanical model that quantifies the impulse of the maneuver. The objective of this work is to understand the fundamental mechanisms of thrust generation in unsteady fast-start swimming.

[1]We acknowledge support of EADS Foundation through the project "Fluids and elasticity in biomimetic propulsion" and of the Chaire Total for RZ as a visiting professor at ESPCI ParisTech.

8:39AM G18.00004 The swimming mechanics of Artemia Salina , A. RUIZ-ANGULO, A.K. RAMOS-MUSALEM, R. ZENIT, Universidad Nacional Autonoma de Mexico — An experimental study to analyze the swimming strategy of a small crustacean (Artemia Salina) was conducted. This animal has a series of eleven pairs of paddle-like appendices in its thorax. These legs move in metachronal-wave fashion to achieve locomotion. To quantify the swimming performance, both high speed video recordings of the legs motion and time-resolved PIV measurements of the induced propulsive jet were conducted. Experiments were conducted for both tethered and freely swimming specimens. We found that despite their small size, the propulsion is achieved by an inertial mechanism. An analysis of the efficiency of the leg wave-like motion is presented and discussed. A brief discussion on the mixing capability of the induced flow is also presented.

8:52AM G18.00005 Fluid elasticity enhances the locomotion of multi-tail swimmers , F.A. GODINEZ, S. GOMEZ, R. ZENIT, Universidad Nacional Autonoma de Mexico, E. LAUGA, University of Cambridge — We conducted experiments on the locomotion of magnetic robots with multiple rigid flagella to evaluate the impact of fluid viscoelasticity on their swimming performance. Each swimmer was composed of a air-filled cylindrical head with a permanent magnet attached at one of its ends. At the other end, two or more rigid helices were glued on the outer surface of the cylinder maintaining the same distance from each other along the periphery and remaining parallel to the rotation axis. The robots were driven by an external magnetic field allowing to vary the swimming speed. Each swimmer was tested in two different fluids with the same shear viscosity: a Newtonian and a Boger fluid. The single-flagellum device showed essentially the same velocity in both fluids. In contrast, multi-flagella robots swam in the Boger fluid at much higher speeds than in the equivalent Newtonian case. These results are discussed in the last of past similar studies.

9:05AM G18.00006 High Speed Tomographic PIV Measurements of Copepod Sensitivity to a Suction-Feeding Predator Mimic , J. YEN, Georgia Tech, D.W. MURPHY, Johns Hopkins University, L. FAN, A. SKIPPER, D.R. WEBSTER, Georgia Tech — Copepods, which sense their fluid environment with long, setae-bearing antennules, often serve as prey to fish. The fluid disturbance created by fish feeding is a combination of a bow wave created by swimming towards the prey with an open mouth and a sudden, high speed flow into the fish's mouth created by suction. The sensitivity and reaction of copepods to the dynamic, high acceleration flow created by a suction feeding fish have not been well explored. In the present study, a suction feeding piscine predator mimic is developed and tested with copepods from a fish-containing (marine) environment (*Calanus finmarchicus*) and with copepods from a fish-less (alpine lake) environment (*Hesperodiaptomus shoshone*). Flow fields created by the impulsive siphon are measured with a high-speed tomographic particle image velocimetry (PIV) system. Escape success and kinematics of the two species are compared. Finally, using volumetric flow measurements, the hydrodynamic signal measured along each copepod's antennules at the time point of escape is compared between species.

9:18AM G18.00007 Thin Layer Sensory Cues Affect Antarctic Krill Swimming Kinematics , A.C. TRUE, D.R. WEBSTER, M.J. WEISSBURG, J. YEN, Georgia Tech — A Bickley jet (laminar, planar free jet) is employed in a recirculating flume system to replicate thin shear and phytoplankton layers for krill behavioral assays. Planar laser-induced fluorescence (LIF) and particle image velocimetry (PIV) measurements quantify the spatiotemporal structure of the chemical and free shear layers, respectively, ensuring a close match to *in situ* hydrodynamic and biochemical conditions. Path kinematics from digitized trajectories of free-swimming *Euphausia superba* examine the effects of hydrodynamic sensory cues (deformation rate) and bloom level phytoplankton patches ($\sim$1000 cells/mL, *Tetraselamis spp.*) on krill behavior (body orientation, swimming modes and kinematics, path fracticality). Krill morphology is finely tuned for receiving and deciphering both hydrodynamic and chemical information that is vital for basic life processes such as schooling behaviors, predator/prey, and mate interactions. Changes in individual krill behavior in response to ecologically-relevant sensory cues have the potential to produce population-scale phenomena with significant ecological implications. Krill are a vital trophic link between primary producers (phytoplankton) and larger animals (seabirds, whales, fish, penguins, seals) as well as the subjects of a valuable commercial fishery in the Southern Ocean; thus quantifying krill behavioral responses to relevant sensory cues is an important step towards accurately modeling Antarctic ecosystems.

9:31AM G18.00008 Quantifying copepod sensing and swimming in unsteady flow fields using time-resolved tomographic PIV + 3D PTV[1] , DEEPAK ADHIKARI, ELLEN LONGMIRE, University of Minnesota — Copepods respond to hydrodynamic disturbances by executing an escape response jump (Buskey et al 2002; Fields and Yen 1996; Kiørboe et al 1999; Strickler and Bal 1973). 3D PTV and tomographic PIV are combined to track the motion of the copepods and the surrounding fluid velocity field respectively, to provide quantitative measure on their sensing and swimming behavior. The measurements are time-resolved to obtain the entire trajectories of copepods. Fluid velocity and velocity gradients are estimated at the location of the copepod by applying a Taylor-series least-square method to the surrounding PIV grid points. Copepod sensing and swimming are analyzed upstream and downstream of a wall-mounted cylinder in cross-flow at Re $\sim$ 930. At the upstream location, when copepods approach the cylinder, they respond by rapidly accelerating (or jumping) away from it. Their jump location suggests that they sense and respond to a range of flow velocity gradients. Preliminary results indicate that copepods in the cylinder wake do not jump frequently as compared to upstream. The velocity gradient thresholds for sensing and range of maximum velocities during jumping will be presented and discussed.

[1]Supported by NSF-IDBR grant #0852875.

9:44AM G18.00009 Investigating the relationship between planform and performance in bio-inspired aquatic propulsion[1] , OLIVER J. BADAOUI, DANIEL B. QUINN, PETER A. DEWEY, Princeton University, ALEXANDER J. SMITS, Princeton University and Monash University — Experiments are conducted to investigate the effects of caudal fin planform shape on the hydrodynamic performance of bio-inspired aquatic propulsors. To isolate the effect of planform shape the surface area of the fins is held constant while the planform shape is systematically varied to incorporate bio-inspired designs that are consistent with those observed in nature. The self-propelled swimming speed and power consumption of heaving flexible panels of varying planform are measured in a stationary water tank. Particle image velocimetry is also employed to better understand the connection between the wake structures produced by the oscillating fins and their performance characteristics. Results are compared and analyzed in an effort to identify specific shape features that lead to a performance benefit or detriment.

[1]Supported by the Office of Naval Research under Program Director Dr. Bob Brizzolara, MURI grant number N00014-08-1-0642.

9:57AM G18.00010 Free swimming of an internally actuated elastic swimmer , PETER YEH, ALPER ERTURK, ALEXANDER ALEXEEV, Georgia Institute of Technology — We use fully coupled three-dimensional simulations to examine the underwater locomotion of an internally powered elastic swimmer. The swimmer is modeled as a thin, rectangular, elastic plate with two sections. The first section is internally powered by an oscillating internal moment that produces bending. The second section, a passive fin, undergoes bending oscillations in response to the actuated section. We measure the forward swimming velocity and performance for our hybrid swimmer. We find that the hybrid swimmer with the passive component swims at a higher velocity than that of a fully actuated one. This is in agreement with experiments involving piezoelectric internally powered swimmers. The experiments have shown that thrust is increased when a passive fin is attached to a fully internally actuated swimmer. We investigate the details of the flow structures and bending pattern of the swimmer and show how they affect the forward motion. The results are useful for designing self-propelling bio-inspired robots with internally powered fins.

Monday, November 25, 2013 8:00AM - 9:57AM –

Session G19 Biofluids: Cellular II - Experimental Studies 310/311 - Juan Carlos Del Alamo, University of California, San Diego

8:00AM G19.00001 Mechanical Response of Red Blood Cells Entering a Constriction , NANCY ZENG, WILLIAM RISTENPART, Dept. Chemical Engineering & Materials Science, University of California, Davis — Most work on RBC dynamic response to hydrodynamic stress has focused on linear velocity gradients. Relatively little experimental work has examined how RBCs respond to pressure driven flow in more complex geometries, such as in an abrupt contraction. Here, we establish the mechanical behaviors of RBCs undergoing a sudden increase in shear stress at the entrance of a narrow constriction. We pumped RBCs through a constriction in an ex vivo microfluidic device and used high speed video to visualize and track the flow behavior of more than 4,000 RBCs. We show that approximately 90% of RBCs undergo one of four distinct modes of motion: stretching, twisting, tumbling, or rolling. Intriguingly, almost 40% of the cells exhibited twisting (rotation around the major axis parallel to the flow direction), a mechanical behavior that is not typically observed in linear velocity gradients. We present detailed statistical analyses on the dynamics of each motion and demonstrate that the behavior is highly sensitive to the location of the RBC within the channel. Finally, we show that the tumbling and rolling motions can be rationalized qualitatively in terms of rigid body rotation, whereas twisting motion cannot, suggesting that twisting is a consequence of the viscoelastic nature of the RBCs.

8:13AM G19.00002 Effect of Varying Fluid Shear Stress on Cancer Stem Cell Viability & Protein Expression[1] , RIA DOMIER, YONGHYUN KIM, DAVID DOZIER, URSULA TRIANTAFILLU, University of Alabama — Cancer stem cells cultured *in vitro* in stirred bioreactors are exposed to shear stress. By observing the effect of shear stress on cancer stem cell viability, laboratory cell growth could be optimized. In addition, metastasized cancer stem cells *in vivo* are naturally exposed to shear stress, a factor influencing stem cell differentiation, while circulating in the bloodstream. Changes in protein expression after exposure to shear stress could allow for identification and targeting of circulating cancer cells. In this study, blood flow through capillaries was simulated by using a syringe pump to inject suspensions of Kasumi-1 leukemia stem cells into model blood vessels composed of PEEK tubing 125 microns in diameter. The Hagen-Poisseuille equation was used to solve for operating flow rates based on specified amounts of shear stress. After exposure, cell counts and viabilities were observed using an optical microscope and proteins were analyzed using Western blotting. It was observed that at a one minute exposure to stress, cell viability increased as the amount of shear was increased from 10 to 60 dynes per square centimeter. Results from this research are applicable to optimization of large-scale stem cell growth in bioreactors as well as to the design of targeted cancer therapies.

[1]Funding from NSF REU grant #1062611 is gratefully acknowledged.

8:26AM G19.00003 Lower limit of shear to induce 2-D protein crystals[1] , JAMES YOUNG, DAVID POSADA, AMIR HIRSA, Rensselaer Polytechnic Institute, JUAN LOPEZ, Arizona State University — Proteins are an essential part of every organism. Protein functionality depends on its structure. In order to utilize the most widely used and powerful technique of X-ray crystallography, the protein must first be crystallized. Crystallization is not a trivial step and success rate is often dismal. One approach is two-dimensional protein crystallization at the air/water interface which entails the binding of protein initially in solution to a ligand that has been spread on the interface to form a monolayer. 2-D crystallization avoids some of the complications of 3-D crystallization such as gravity. It also reduces the amount of protein needed by 3 orders of magnitude. Here we quantify the level of interfacial shearing needed to enable crystals to be formed at protein surface concentrations lower than those required in a quiescent system. A phase diagram is presented delineating the required shear rate for a given surface pressure. In addition, surface shear viscosity is demonstrated to be a sensitive macroscopic probe for the in-situ detection of flow-induced crystals.

[1]supported by NSF-CBET

8:39AM G19.00004 Are endothelial cell bioeffects from acoustic droplet vaporization proximity dependent? , ROBINSON SEDA, DAVID LI, J. BRIAN FOWLKES, JOSEPH BULL, University of Michigan — Acoustic droplet vaporization (ADV) produces gas microbubbles that provide a means of selective occlusion in gas embolotherapy. Vaporization and subsequent occlusion occur inside blood vessels supplying the targeted tissue, such as tumors. Theoretical and computational studies showed that ADV within a vessel can impart high fluid mechanical stresses on the vessel wall. Previous in vitro studies have demonstrated that vaporization at an endothelial layer may affect cell attachment and viability. The current study is aimed at investigating the role of vaporization distance away from the endothelial layer. HUVECs were cultured in OptiCell^{TM}chambers until reaching confluence. Dodecafluoropentane microdroplets were added, attaining a 10:1 droplet to cell ratio. A single ultrasound pulse (7.5 MHz) consisting of 16 cycles ($\sim$ 2 μs) and a 5 MPa peak rarefactional pressure was used to produce ADV while varying the vaporization distance from the endothelial layer (0 μm, 500 μm, 1000 μm). Results indicated that cell attachment and viability was significantly different if the distance was 0 μm (at the endothelial layer). Other distances were not significantly different from the control. ADV will significantly affect the endothelium if droplets are in direct contact with the cells. Droplet concentration and flow conditions inside blood vessels may play an important role. This work was supported by NIH grant R01EB006476.

8:52AM G19.00005 Fabrication of hydrogel substrates with stiffness step variations using controlled surface wettability , MD. MAHMUDUR RAHMAN, DONGHEE LEE, SANGJIN RYU, University of Nebraska-Lincoln — Living cells can respond to changes in the stiffness of the surrounding matrix. Well-known examples include the durotaxis of motile cells and the stiffness-dependent differentiation of stem cells. Such mechanobiological behaviors of living cells have been investigated on hydrogel substrates of which the compliance is either static or varying in one direction. Although various techniques have been developed to fabricate hydrogel substrates of controllable stiffness distributions, however, the fabricated substrates have only hydrogel regions of varying stiffness, lacking regions of static stiffness. Therefore, it has been difficult to compare cells' responses to static stiffness and varying stiffness under the same culture condition. Thus, we aim to fabricate polyacrylamide gel substrates consisting of alternating regions of static stiffness and stiffness gradient. For controlled positioning of gel solutions with different relative concentrations of acrylamide and the crosslinker, we generated superhydrophilic regions surrounded by hydrophobic barriers on glass and then filled the regions with the gel solutions. When sandwiched by another glass surface, the gel solutions experienced limited mixing only at interfaces, which created stiffness gradients between static stiffness regions.

9:05AM G19.00006 Turbulent unmixing: how marine turbulence drives patchy distributions of motile phytoplankton , WILLIAM DURHAM, Oxford University, ERIC CLIMENT, Institut de Mecanique des Fluides, MICHAEL BARRY, MIT, FILIPPO DE LILLO, GUIDO BOFFETTA, Universita di Torino, MASSIMO CENCINI, Consiglio Nazionale delle Ricerche, ROMAN STOCKER, MIT — Centimeter-scale patchiness in the distribution of phytoplankton increases the efficacy of many important ecological interactions in the marine food web. We show that turbulent fluid motion, usually synonymous with mixing, instead triggers intense small-scale patchiness in the distribution of motile phytoplankton. We use a suite of experiments, direct numerical simulations of turbulence, and analytical tools to show that turbulent shear and acceleration directs the motility of cells towards well-defined regions of flow, increasing local cell concentrations more than ten fold. This motility-driven 'unmixing' offers an explanation for why motile cells are often more patchily distributed than non-motile cells and provides a mechanistic framework to understand how turbulence, whose strength varies profoundly in marine environments, impacts ocean productivity.

9:18AM G19.00007 Getting into the flow: Red cells go on a roll, two-component vesicles swing , ANNIE VIALLAT, JULES DUPIRE, KAMEL KHELLOUFI, AL HAIR AL HALIFA, Aix Marseille Université, CNRS UMR7333, lab. Adhesion and Inflammation, Inserm, ADHESION AND INFLAMMATION TEAM — Red blood cells are soft capsules. Under shear flow, their two known motions were "tumbling" and "swinging-tank treading," depending on cell mechanics and flow conditions. We reveal new wobbling regimes, among which the "rolling" regime, where red cells move as wheels on a road. We show, by coupling two video-microscopy approaches providing multi-directional cell pictures that the orientation of cells flipping into the flow is determined by the shear rate. Rolling permits to avoid energetically costly cellular deformations and is a true signature of the cytoskeleton elasticity. We highlight two transient dynamics: an intermittent regime during the "tank-treading-to-flipping" transition and a Frisbee-like "spinning" regime during the "rolling-to-tank-treading" transition. We find that the biconcave red cell shape is very stable under moderate shear stresses, and we interpret this result in terms of shape memory and elastic buckling. Finally, we generate lipid vesicles with a shape memory by using two lipids with different bending rigidities. These vesicles swing in shear flow similarly to red blood cells but their non-axisymmetric stress-free shape changes the periodicity of the motion and induces specific features.

9:31AM G19.00008 Platelet transport in microchannels , MATHILDE REYSSAT, ANNE LE GOFF, ESPCI/Gulliver, Paris, France, ANTOINE BLIN, PlatOD, JUSTINE PUJOS, ESPCI/Gulliver, Paris, France, AURÉLIE MAGNIEZ, PlatOD, DOMINIQUE BARUCH, INSERM, U765, Paris, France — Blood platelets are small enucleated cells responsible for the arrest of bleeding. These cells have the ability to tether and translocate on injured vascular endothelium, thanks to a specific interaction between a receptor of their membrane and a protein expressed by the cells composing the inner wall of the vessel, the von Willebrand factor (VWF). Others cells have such abilities of rolling. Leucocytes, for example, translocate on surface due to a specific interaction between selectin molecules and their respective glycoprotein ligands. These kinds of cells present two modes of transport: they can either be advected by the flux, or translocate on surfaces due to specific ligand-receptor interactions. Our work consists first in studying experimentally the transport of platelets along a microchannel and then in modeling this particular cell transport. Due to these two modes of transport along a channel, platelets adhering to the surface are not equally distributed along the channel axis. We describe the evolution of the density of platelets with time and distance.

9:44AM G19.00009 Characterization of Intracellular Streaming and Traction Forces in Migrating Physarum Plasmodia , SHUN ZHANG, MAE Dept, UC San Diego, RUEDI MEILI, Biology Dept, UC San Diego, ROBERT D. GUY, Math Dept, UC Davis, JUAN C. LASHERAS, JUAN C. DEL ALAMO, MAE Dept, UC San Diego — Physarum plasmodium is a model organism for cell migration that exhibits fast intracellular streaming. Single amoebae were seeded and allowed to move on polyacrilamide gels that contained 0.2 μm fluorescent beads. Joint time-lapse sequences of intracellular streaming and gel deformation were acquired respectively in the bright and fluorescent fields of a confocal microscope. Images were analyzed using particle image velocimetry (PIV) algorithms, and the traction stresses applied by the amoebae on the surface were computed by solving the elastostatic equation for the gel. These measurements provide, for the first time, a joint characterization of intracellular mass transport and the forces applied on the substrate of motile amoeboid cells with high resolution in both time and space, enables a through study about the locomotive mechanism and the relation between intracellular flow and traction stress, shedding light on related biomimetic research. The results reveal a pronounced auto-oscillation character in intracellular flow, contact area, centroid speed and strain energy, all with the same periodicity about 60 seconds. Adhesion sites are found to be almost stationary while a traction wave propagates from the tail to the anterior region in each cycle.

Monday, November 25, 2013 8:00AM - 10:10AM –

Session G20 Boundary Layers V: Compressible and Thermal 315 - Farzad Mashayek, University of Illinois at Chicago

8:00AM G20.00001 A quantitative theory for the mean velocity distribution of compressible ramp flow , WEI-TAO BI, BIN WU, HONG-YUE ZOU, Peking University, XIN-LIANG LI, Institute of Mechanics, CAS, FAZLE HUSSAIN, Texas Tech University, ZHEN-SU SHE, Peking University — The flow induced by a compression ramp is of practical importance as a typical flow in the intake of a scramjet engine, yet no quantitative theory is available. This study proposes a quantitative theory for the mean velocity profile (MVP) of the compression ramp flow, based on a multi-layer description of turbulent boundary layers. Application of the theory on the direct numerical simulation (DNS) data shows that the mixing length function in the boundary layer after the reattachment point has a five-layer structure. A formula is given for the streamwise MVP, in very good agreement with the DNS data. Variation of the parameters in the formula with the spatial position is measured and discussed. These results further support the validity of the Structural Ensemble Dynamics approach to a wide class of wall-bounded flows, and a new modeling strategy for engineering computation of complex supersonic flows.

8:13AM G20.00002 Turbulence Structure and Wall Signature in Hypersonic Boundary Layer , YIN CHIU KAN, PINO MARTIN, University of Maryland, College Park — We will investigate the turbulence structure from direct numerical simulation (DNS) data of Mach 3 and Mach 7 turbulent boundary layers. In particular, we will use linear stochastic estimation to provide evidence of hairpin structures, examine the character of coherent structures statistically and instantaneously, as well as their wall signatures. In addition, we will use a spatio-temporal pattern finding process to track multiple packets evolutions concurrently.

8:26AM G20.00003 Thermal boundary condition effects on compressible turbulent boundary layers[1] , IZAAK BEEKMAN, Princeton University, PINO MARTIN, University of Maryland — Numerous questions about the physics of compressible boundary layers, and their modeling remain open. While Morkovin's hypothesis has proven remarkably robust for zero pressure gradient, smooth wall, compressible, turbulent boundary layers, accounting correctly for thermal energy transport and its impact on the density and momentum fields remains challenging. We use spatially developing DNS data over strongly and weakly adiabatic walls at Mach 3 and Mach 7. The strongly adiabatic boundary condition further stresses common assumptions of weak direct compressibility and weak total temperature fluctuations. We observe non-trivial differences between the two cases. The simulations are performed at $Re_\tau \approx 500$ on very large domains in the streamwise and spanwise directions, approximately 50 by $10\delta_{inlet}$, with a rescaling method providing the inflow. We examine the effects of this boundary condition on common scaling laws, temperature-velocity relations, and suggest improvements, where possible. A dimensionless parameter is proposed, the "fluctuation Nusselt number," to quantify the impact of the wall material for laboratory and engineering flows and relate it to these idealized, numerical boundary conditions.

[1]Supported by AFOSR grant AF/9550-10-1-0535 STW 21 - Revitalization of the hypersonics testing and evaluation workforce.

8:39AM G20.00004 Interaction of a Mach 2.25 turbulent boundary layer with a fluttering panel using direct numerical simulation[1] , DANIEL BODONY, CHRISTOPHER OSTOICH, PHILIPPE GEUBELLE, University of Illinois at Urbana-Champaign — The interaction between a thin metallic panel and a Mach 2.25 turbulent boundary layer is investigated using a direct numerical simulation approach for coupled fluid-structure problems. The solid solution uses a finite-strain, finite-deformation formulation, while the direct numerical simulation of the boundary layer uses a finite-difference compressible Navier-Stokes solver. The initially laminar boundary layer contains low amplitude unstable eigenmodes that grow in time and excite traveling bending waves in the panel. As the boundary layer transitions to a fully turbulent state, with $Re_\theta \approx 1200$, the panel's bending waves coalesce into a standing wave pattern exhibiting flutter with a final amplitude approximately 20 times the panel thickness. The corresponding panel deflection is roughly 25 wall units and reaches across the sonic line in the boundary layer profile. Once it reaches a limit cycle state, the panel/boundary layer system is examined in detail where it is found that turbulence statistics, especially the main Reynolds stress $-\langle u'v' \rangle$, appear to be modified by the presence of the compliant panel, the effect of which is forgotten within one integral length downstream of the panel.

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

8:52AM G20.00005 Acoustic Radiation from High-Speed Turbulent Boundary Layers[1] , LIAN DUAN, Missouri University of Science and Technology, MEELAN CHOUDHARI, NASA Langley Research Center — Direct numerical simulations (DNS) are used to examine the pressure fluctuations generated by a high-speed turbulent boundary layer with nominal freestream Mach number of 6 and Karman number of $Re_\tau \approx 464$. The emphasis is on comparing the primarily vortical pressure signal at the wall with the acoustic freestream signal under higher Mach number conditions. Moreover, the Mach-number dependence of pressure signals is investigated by comparing the current results with those of a supersonic boundary layer at Mach 2.5 and $Re_\tau \approx 510$. It is found that the freestream pressure intensity exhibits a strong Mach number dependence, irrespective of whether it is normalized by the mean wall shear stress or by the mean pressure. Spectral analysis shows that both the wall and freestream pressure fluctuations of the Mach 6 boundary layer have enhanced energy content at high frequencies. The computed Mach-number dependence of the acoustic field, including radiation intensity, directionality, and convection speed, is consistent with trends in measurements. The numerical database is used to understand the acoustic source mechanisms for both adiabatic and cold wall configurations.

[1]Supported by NASA

9:05AM G20.00006 Effects of radiation in turbulent boundary layers: Analysis of the mean temperature profile , RONAN VICQUELIN, YUFANG ZHANG, OLIVIER GICQUEL, JEAN TAINE, Ecole Centrale Paris, CNRS EM2C — Direct numerical simulations fully coupled with radiative energy transfer in a turbulent channel flow have been performed for different temperature, optical thickness (pressure) and wall emissivity conditions. Radiation is treated from the CK approach and a Monte Carlo transfer method. Analysis of the results shows that, beside an additional wall radiative flux, the structure of the mean temperature field and the wall conductive flux often strongly differ from results without radiation. It is found that gas-gas and gas-wall radiation interactions generate antagonist effects. The first one tends to increase wall conductive flux while the second one to decrease it. Classical wall log-laws for temperature are therefore strongly modified by the global radiation effects. Many conditions encountered in applications are discussed. The observed modifications depend on all the set of conditions (temperature level, wall emissivity, pressure, Reynolds number), i.e. on the relative magnitudes of radiation gas-gas and gas-wall phenomena and of global radiation flux and conductive flux without radiation.

9:18AM G20.00007 A wall model for LES accounting for radiation effects , RONAN VICQUELIN, YUFANG ZHANG, OLIVIER GICQUEL, JEAN TAINE, Ecole Centrale Paris, CNRS EM2C — In several conditions, radiation can modify the temperature law in turbulent boundary layers. In order to predict such an effect and the corresponding change in conductive heat flux at the wall, a new wall model for large eddy simulation (LES) is proposed. The wall model describes the inner boundary layer which cannot be resolved by the LES. The radiative power source term is calculated from an analytical expression of the intensity field within the inner layer. In the outer layer, wall stress and conductive heat flux predicted by the wall model are fed back to the LES which is coupled to a reciprocal Monte-Carlo method to account for radiation. Several mixing-length models and turbulent Prandtl number formula are investigated. Then, the level of accuracy of the discretized radiation analytical model is investigated. Finally, fully coupled results are compared with Direct Numerical Simulation/Monte-Carlo results of turbulent channel flows at different Reynolds number, wall temperature and pressure conditions. The proposed wall model greatly improves the accuracy of the predicted temperature profiles and wall conductive heat fluxes compared to approaches without radiation accounted for in the inner layer.

9:31AM G20.00008 Experimental investigation of thermally stable turbulent boundary layers , ALEXANDER J. SMITS, Princeton University, Monash University, OWEN WILLIAMS, TRISTEN HOHMAN, Princeton University, TYLER VAN BUREN, Rensselaer Polytechnic Institute — Thermally stable turbulent boundary layers are prevalent in the polar regions and instrumental in determining surface heat fluxes. At present, theoretical treatments of such flows have been found to be inaccurate. Experiments were thus conducted to gain further insight into changes in turbulent structure and corresponding statistics under stable conditions. Isothermal and constant heat flux boundary conditions were investigated as well as smooth and rough surfaces. PIV was used to examine the velocity field, and a thermocouple rake was used to measure the mean temperature profile. Under particular investigation are (1) the existence of a critical Richardson number at which turbulence was strongly suppressed and whether this was influenced by the surface roughness condition, and (2) the effects of increased stratification on the hairpin vortex structure and its organization into packets. This work was made possible by support received through Princeton University's Grand Challenges-Energy program, supported by the Thomas and Stacey Siebel Foundation.

9:44AM G20.00009 Turbulent thermal boundary layers subjected to severe acceleration , GUILLERMO ARAYA, LUCIANO CASTILLO, Texas Tech University, Lubbock, TX — Favorable turbulent boundary layers are flows of great importance in industry. Particularly, understanding the mechanisms of quasi-laminarization by means of a very strong favorable streamwise pressure gradient is indeed crucial in drag reduction and energy management applications. Furthermore, due to the low Reynolds numbers involved in the quasi-laminarization process, abundant experimental investigation can be found in the literature for the past few decades. However, several grey zones still remain unsolved, principally associated with the difficulties that experiments encounter as the boundary layer becomes smaller. In addition, little attention has been paid to the heat transfer in a quasi-laminarization process. In this investigation, DNS of spatially-developing turbulent thermal boundary layers with prescribed very strong favorable pressure gradients (K=4x10-6) are performed. Realistic inflow conditions are prescribed based on the Dynamic Multi-scale Approach (DMA) [Araya et al. JFM, vol. 670, pp. 581-605, 2011]. In this sense the flow carries the footprint of turbulence, particularly in the streamwise component of the Reynolds stresses.

9:57AM G20.00010 ABSTRACT WITHDRAWN –

Monday, November 25, 2013 8:00AM - 10:10AM –

Session G21 Turbulence: Simulations III - DNS and LES II 316 - Robert Moser, University of Texas

8:00AM G21.00001 Progress and opportunities in direct numerical simulations at the next higher resolution[1] , P.K. YEUNG, Georgia Institute of Technology, K.R. SREENIVASAN, New York University — In recent years, many researchers in the turbulence community have been able to exploit the steady advancement of computing power to advance our understanding of turbulence, including new parameter ranges and the effects of coupling with other physical processes. However it is remarkable that, the "record" grid resolution of 4096^3, first achieved just over 10 years ago (Kaneda *et al.*, *Phys. Fluids* 2003) still stands in the literature of the field. In this talk, we will present preliminary results from an 8192^3 simulation of turbulence on a periodic domain, carried out using 262144 CPU cores on the *Blue Waters* supercomputer under the NSF Track 1 Petascale Resource Allocations program. Since a simulation at this magnitude is still extremely expensive, and the resources required are not easily secured, very careful planning and very aggressive efforts at algorithmic enhancement are necessary (which we will also briefly discuss). This new simulation isexpected to allow us to probe deeply into fundamental questions such as intermittency at the highest Reynolds numbers and the best possible resolution of the small scales at the current limit of computing power available.

[1]Supported by NSF Grant ACI-1036170.

8:13AM G21.00002 Backward tracking for the study of turbulent dispersion in direct numerical simulations over a range of Reynolds numbers[1] , D. BUARIA, P.K. YEUNG, Georgia Tech, B.L. SAWFORD, Monash Univ., Australia — The dispersive character of turbulence is well known, and readily observed through, for example, the increase with time of the mean separation between fluid particle pairs in a Lagrangian framework. Usually, in both direct numerical simulations (DNS) and laboratory experiments, a population of fluid particles is tracked forward in time from specified initial conditions. However, from a modeling perspective, it is more important to track the particles backwards, which would help address questions about the dynamical origins of a patch of contaminant material, or a highly convoluted multi-particle cluster. In this talk we present numerical results on backward statistics obtained by sampling particle pairs of desired separation at the final time of a relatively long DNS run. The calculation essentially involves processing large datasets consisting of the complete time history of position, velocity and velocity gradients along the trajectories. Promising results on both forward and backward dispersion from up to 16 million particles have been obtained over time intervals spanning the ballistic, inertial, and diffusive ranges. This approach will allow us to study backward dispersion and relate Lagrangian studies to scalar mixing, at Taylor-scale Reynolds numbers up to 1000.

[1]Supported by NSF Grant CBET-1235906

8:26AM G21.00003 A Web-Services accessible database for channel flow turbulence at Re_τ=1000[1] , J. GRAHAM, K. KANOV, E. GIVELBERG, R. BURNS, G. EYINK, A. SZALAY, C. MENEVEAU, Johns Hopkins University, M.K. LEE, N. MALAYA, R.D. MOSER, University of Texas — In this presentation we describe a new public database archiving a DNS data set of the space-time evolution of fully developed channel flow at $Re_\tau = 1000$. The database will contain data from a DNS of channel flow with domain-size of $8\pi \times 2 \times 3\pi$, at the resolution of 2048×512×1536, with 2048 time-frames of velocity and pressure fields spanning about a flow-through time scale. After simulation, the data are ingested into the database cluster using a space-filling Morton-curve to index the computational space uniformly, and also to organize data partition and distribution. The database system allows users access and to process the data remotely through an interface based on the Web-Service model. Users are thus able to perform numerical experiments on the high-resolution DNS data using least capable desktop computers. Test calculations are performed to illustrate the usage of the system and to verify the correctness of the data. Construction of the database also involves developments of MPI-DB, a new tool to facilitate coupling of parallel simulations and databases.

[1]Support provided, among others, from the National Science Foundation CDI-II grant CMMI-0941530.

8:39AM G21.00004 Estimating Uncertainties in Statistics Computed from DNS[1] , NICHOLAS MALAYA, TODD OLIVER, RHYS ULERICH, ROBERT MOSER, University of Texas at Austin — Rigorous assessment of uncertainty is crucial to the utility of DNS results. Uncertainties in the computed statistics arise from two sources: finite sampling and the discretization of the Navier-Stokes equations. Due to the presence of non-trivial sampling error, standard techniques for estimating discretization error (such as Richardson Extrapolation) fail or are unreliable. This talk provides a systematic and unified approach for estimating these errors. First, a sampling error estimator that accounts for correlation in the input data is developed. Then, this sampling error estimate is used as an input to a probabilistic extension of Richardson extrapolation in order to characterize the discretization error. These techniques are used to investigate the sampling and discretization errors in the DNS of a wall-bounded turbulent flow at $Re_\tau = 180$. We will show a well-resolved DNS simulation which, for the centerline velocity, possesses 0.02% sampling error and discretization errors of 0.003%. These results imply that standard resolution heuristics for DNS accurately predict required grid sizes.

[1]This work is supported by the Department of Energy [National Nuclear Security Administration] under Award Number [DE-FC52-08NA28615].

8:52AM G21.00005 Direct numerical simulation for incompressible channel flow at $Re_\tau = 5200$[1] , MYOUNGKYU LEE, NICHOLAS MALAYA, ROBERT D. MOSER, The University of Texas at Austin — High-resolution direct numerical simulation (DNS) of wall-bounded canonical channel flow at $Re_\tau = 5200$ is performed. The computational domain is $8\pi\delta \times 2\delta \times 3\pi\delta$ with $10240 \times 1536 \times 7680$ grid points in streamwise(x), wall-normal direction(y), and spanwise(z) directions, respectively. Fourier spectral method(x, and z) and B-splines(y) are used for the the computation of derivatives. In this presentation we demonstrate that the simulation exhibits several features of high Reynolds number wall-bounded turbulence. The value of von Kármán constant appears to be $\kappa = 0.384$ in the region of y^+=300 $\sim$ y=0.2δ where the mean velocity profile shows logarithmic variation. Also, distinct inner(λ_x^+= 800, λ_z^+= 120) and outer(λ_x= 8δ, λ_z= δ) peaks in one-dimensional premultiplied spectra of the velocity variance are observed. Finally, the k_x^{-1} region is observed in the range of y^+= 120 $\sim$ 150 and k_x= 6 $\sim$ 10.

[1]This work is supported by NSF PetaApps grants: OCI-0749223 and NSF PRAC Grant 0832634.

9:05AM G21.00006 LES of radial wall jets over smooth and rough surfaces , RAYHANEH BANYASSADY, UGO PIOMELLI, Queen's University — Large-eddy simulations of turbulent radial wall jets were conducted over both smooth and rough surfaces ($5 < k_s^+ < 70$) at the Reynolds number of $40,000$ (based on the bulk velocity and diameter of the impinging jet). The roughness elements are represented using a virtual sandpaper model and an immersed-boundary method (IBM). The results of the simulation are validated with available experimental data and also compared to our previous plane wall-jet simulations. The radial wall jets spread faster compared to the plane ones due to expansion in two directions. However, the effects of roughness are similar and mostly confined to the inner layer of the wall jet. Roughness enhances isotropy in this region. The damping of wall normal velocity fluctuations by the wall decreases over the rough wall which increases the penetration height of the wall jet. In the outer layer, normalizing with maximum velocity and wall jet half-height, $y_{1/2}$, can collapse mean velocity and Reynolds stresses profiles. There is no significant structural difference at the same non-dimensional height from the wall ($y/y_{1/2}$) over the smooth and the rough wall. An analysis of the effect of surface roughness on the instantaneous flow structures is also presented.

9:18AM G21.00007 Large-eddy simulation of turbulent dispersion from a localized source in a build-up environment , BING-CHEN WANG, MOHAMMAD SAEEDI, Univ. of Manitoba — Turbulent dispersion from a continuous ground-level point-source within an array of 16x16 cubes has been simulated using wall-modelling large-eddy simulations. The major challenges associated with this problem involve obtaining a deeper understanding of the interaction of the dynamically evolving flow structures with the complex boundary conditions, coupling of the momentum and scalar transport processes, and a high Reynolds number tested for an modeled urban atmospheric boundary layer (Re=12,005 based on the free stream velocity and obstacle height). A fully-parallelized in-house computer code was used for performing the simulation. An advanced dynamic nonlinear model (DNM) and dynamic full linear eddy diffusivity model (DFLTDM) have been used for closure of the filtered momentum and scalar transport equations, respectively. A non-equilibrium thin boundary-layer wall model is applied to all solid surfaces. Inlet boundary conditions based on solid grids have also been investigated in order to generate high turbulence levels typical for an approaching urban atmospheric boundary-layer flow. The predicted results for the flow and concentration field have been thoroughly validated against a set of high-quality water-channel measurement data.

9:31AM G21.00008 Analysis of effective eddy viscosity in DNS results of stratified turbulence , SINA KHANI, MICHAEL L. WAITE, Department of Applied Mathematics, University of Waterloo — In order to perform large-eddy simulation (LES) of stratified turbulence, subgrid-scale (SGS) models are necessary to represent the effects of small scales on large scale motions. Since the inertial subrange of stratified turbulence is anisotropic for scales larger than the Ozmidov scale, isotropic SGS models do not seem to be the proper approach for LES of stratified turbulence. In this talk, direct numerical simulations of decaying stratified turbulence are analyzed to investigate the effective eddy viscosity in the presence of stratification. The results are studied under different Reynolds numbers, stratifications, and test cutoffs. It is shown that the presence of stratification causes a non-local horizontal energy transfer between large and small horizontal scales, which is not seen for the unstratified case. These results suggest the idea of using different eddy viscosities in the horizontal and vertical directions for scales larger than the Ozmidov scale, as is often done in large-scale atmospheric models, in which stratification is important. Overall, isotropic SGS models should be modified for use in LES of stratified turbulence.

9:44AM G21.00009 ABSTRACT WITHDRAWN –

9:57AM G21.00010 The k^{-2} spectrum in decaying magnetohydrodynamic turbulence[1] , VASSILIOS DALLAS, ALEXANDROS ALEXAKIS, Laboratoire de Physique Statistique, Ecole Normale Superieure, Paris, NON-LINEAR PHYSICS TEAM — We investigate the origins of the k^{-2} spectrum in a decaying Taylor-Green magnetohydrodynamic flow with zero large scale magnetic flux. So far a possible candidate for this scaling exponent has been the weak turbulence phenomenology. From our numerical simulations, we observe that current sheets in this flow are formed in regions of magnetic discontinuities. Based on this observation and by studying the influence of the current sheets on the energy spectrum, using a filtering technique, we demonstrate that magnetic discontinuities are responsible for the -2 power law scaling of the energy spectra in this flow. We also show that initial strong correlations between the velocity and the magnetic field exhibit a k^{-2} spectrum at the peak of dissipation of decaying MHD turbulence. Therefore, the presence of a clear k^{-2} spectrum due to strong current sheets could imply lack of universality in decaying MHD turbulence. However, our highest resolution simulations (2048^3) indicate that we have classes of universality at moderate Reynolds numbers and possibly a universal power law in the high Reynolds number limit.

[1]V.D. acknowledges the financial support from EU-funded Marie Curie Actions FP7-PEOPLE-2011-IEF (Project No. 299973).

Monday, November 25, 2013 8:00AM - 9:57AM –

Session G22 Turbulent Mixing I: Scalar Mixing 317 - Rodney Fox, Iowa State University

8:00AM G22.00001 Measurements of the relative diffusion of a passive scalar plume in sheared turbulence[1] , CHRISTINA VANDERWEL, STAVROS TAVOULARIS, University of Ottawa — A neutrally buoyant aqueous solution of Rhodamine 6G fluorescent dye was injected isokinetically from a thin tube into fully developed, uniformly sheared, nearly homogeneous, turbulent flow in a water tunnel. The turbulent Reynolds number was $Re_\lambda \approx 150$. Instantaneous dye concentration variations in several planes normal to the flow were mapped by means of planar laser-induced fluorescence. Mean relative concentration maps were determined by ensemble averaging the instantaneous maps, each shifted to a common centre of mass. These maps could be fitted well by a 2D Gaussian function. The distance-neighbour function was also determined in each plane from mean planar autocorrelation maps and its shape was found to be comparable to analytical expressions by Richardson and by Batchelor. The relative plume width, defined as the standard deviation of the mean relative concentration map, was approximately equal to $1/\sqrt{2}$ times the standard deviation of the distance-neighbour function. The relative plume width remained within the inertial subrange and its streamwise growth rate was consistent with Richardson's 4/3 Law. The estimated value of Richardson's constant was roughly 0.1.

[1]Supported by NSERC

8:13AM G22.00002 Endwall Vortex Effects on Turbulent Dispersion of Film Coolant in a Turbine Vane Cascade[1] , SAYURI D. YAPA, CHRISTOPHER J. ELKINS, JOHN K. EATON, Stanford University — Turbine flows include strong secondary flows due to flow turning. The dominant flow feature is the passage vortex, located in the corner between the endwall and the suction surface of the airfoil. This vortex may have a strong effect on scalar transport in the turbine wake. Experiments were conducted to examine the dispersion of coolant emitted along the trailing edge of the airfoil. 3D velocity and concentration measurements were made using magnetic resonance imaging to study turbulent mixing in a realistic film-cooled nozzle vane cascade. The passage vortex has large effects on the flow features in the vane wake and on coolant mixing. A shear layer is created on the vane's suction side and interacts with the passage vortex after shedding from the trailing edge. The resulting vortex pattern forces the coolant jet into a highly distorted shape. A key question is how this distortion affects the turbulent diffusion of coolant. The 3D MRI-based velocity and concentration measurements allows for estimation of turbulent diffusivity. Control volumes are defined using a streamtube that is defined beginning just downstream of the trailing edge. The turbulent diffusivity is determined by integrating the Reynolds-averaged advection-diffusion equation over these control volumes.

[1]This work was sponsored by the Army Research Office and General Electric.

8:26AM G22.00003 Transfer of passive scalar variance in decaying grid turbulence with a mean scalar gradient[1] , LUMINITA DANAILA, CORIA CNRS UMR 6614, Université de Rouen, 77801 Saint Etienne du Rouvray, France, LAURENT MYDLARSKI, Department of Mechanical Engineering, McGill University, Montréal, Canada — The present work focuses on the mixing of a passive scalar for $Sc = 0.7$ (Sc is the Schmidt number) in decaying, homogeneous isotropic turbulence, where the scalar fluctuations are produced via a large-scale, mean scalar gradient. The overall philosophy is *to understand and predict the scalar behaviour, when the velocity field is known*. Of particular interest is the transfer of scalar variance and its comparison with that of the kinetic energy. The experimental evidence suggests that the scalar variance transfer is closer to the asymptotic value of $4/3$ than its kinetic energy counterpart. This behaviour is explained analytically by modelling the scalar variance transfer as a function of the scalar variance at scale r, and a characteristic time resulting from the strain effected by a range of scales of sizes slightly larger than r, up to r itself. This model is consistent with the experimental data, measured in grid turbulence with a mean scalar gradient, over a relatively wide range of Reynolds numbers (R_λ up to 600). We highlight the fact that the cascade mechanism of the scalar variance appears to be insensitive to whether the scalar field is dominated by decay or by production.

[1]Support for this work was provided by the CORIA, CNRS, Université de Rouen, ANR and NSERC.

8:39AM G22.00004 Turbulent transport and mixing of a passive scalar in a confined liquid wake , JAMES HILL, KATRINE NILSEN, BO KONG, RODNEY FOX, MICHAEL OLSEN, Iowa State University — Turbulent mixing and transport of a passive scalar has been studied in a confined rectangular liquid wake at Reynolds number 37,500. Large-eddy simulations (LES) of transport and mixing of the passive scalar were performed and the results were compared to velocity and concentration data from simultaneous particle image velocimetry (PIV) and planar laser induced fluorescence (PLIF) measurements. Single-point statistics of velocity and concentration from the LES were validated by the experimental data. Two-point spatial correlations of turbulent velocity, passive scalar, and joint velocity-scalar fields were computed from both simulation and experimental data and the results were compared. In this way information was obtained about LES' ability to predict the important coherent structures of the flow and their contribution to the scalar transport. The simultaneous PIV/PLIF data also provided the opportunity to evaluate turbulent fluxes, turbulent Schmidt number and two components of the turbulent diffusivity tensor. Comparison of these quantities between simulation and experiment provides important insight into the turbulent transport processes of the wake and how LES performs in predicting these. The simulation results showed overall good agreement with the experimental data.

8:52AM G22.00005 Statistical Investigation of Turbulent Mixing by Means of Turbulent Line Segments , MICHAEL GAUDING, NORBERT PETERS, RWTH Aachen University, INSTITUT FUER TECHNISCHE VERBRENNUNG, JARA-HPC TEAM — We examine the turbulent mixing of a passive scalar with imposed mean gradient. The Taylor microscale based Reynolds number varies between 85 and 530. A straight line through the turbulent field of a passive scalar ϕ is decomposed into piece-wise monotonously increasing or decreasing segments. These so called turbulent line segments (TLS) start at a local minimum point and end at a local maximum point or vice versa and are parameterized by the distance ℓ between the extreme points and by the corresponding scalar difference $\Delta\phi$. The implication is that TLS, whose mean length is about ten times the Kolmogorov length, characterize the dynamic process of scalar-energy dissipation. Firstly, we examine the joint distribution function of $\Delta\phi$ and ℓ and define the gradient $\Delta\phi/\ell$ of TLS. This helps to understand cliff-ramp structures as we can show at which length scale large gradients arise. Based on a statistical approach we can further relate the mean gradient to the local gradient and can examine the scaling of the kurtosis of the local gradient with the Reynolds number. Secondly, we define a structure function based on TLS, that relates the extreme points and calculate the scaling exponents. The result is compared with the KOC-theory.

9:05AM G22.00006 Turbulent generation of scalar covariance between two initially distant scalars: implications for enhanced mixing and reaction , MICHAEL SOLTYS, FARROKH SHOAEI, JOHN CRIMALDI, University of Colorado — Mixing and reaction between two scalars initially separated by scalar-free ambient fluid is important in problems ranging from ecology to engineering, but is relatively unstudied compared to the more common topology where the two scalars initially share a material interface. We use a two-channel PLIF system in a laboratory flume to quantify the instantaneous spatial structure of two independent scalars released from laterally separate locations in homogeneous grid turbulence. Local reaction rates in the low-Damkohler limit can then be computed. We demonstrate that the two passive scalars selectively aggregate in attracting regions of the turbulent flow, as quantified by streamwise development of positive scalar covariance. A decomposition of the total reaction into mean and instantaneous contributions reveals that the relative contributions depend strongly on streamwise location. Our results demonstrate that over 80% of the downstream reaction is associated with the scalar covariance produced by instantaneous flow processes, such that the total reaction greatly exceeds that predicted my mean processes alone.

9:18AM G22.00007 Statistical and Visual Analysis of Conserved Scalar Mixing Dynamics in Turbulent Jets Using kHz-Rate Imaging , MICHAEL PAPAGEORGE, FREDERIK FUEST, JEFFRET SUTTON, Ohio State University — The objective of this work is to examine the space-time dynamics of conserved scalar transport and mixing in gas-phase, turbulent jets utilizing kHz-rate, planar laser diagnostics. This research is facilitated by the High-Energy Pulse-Burst Laser System (HEPBLS) at Ohio State, which is capable of delivering high-energy ($\sim$ 1 J) pulses at 532 nm at repetition rates of 10 kHz and higher. With this system, time-resolved 2-D scalar mixing fields are acquired with high signal-to-noise ratios. In this study Rayleigh scattering from a propane jet issuing into a low-speed air co-flow was used to measure mixture fraction at Re = 10,000 to 30,000 at axial locations of x/D = 10 to 40. Single- and multi-point time statistics are employed to gain a better understanding of the dynamics of large-scale features. Single point auto-correlations are used to calculate the integral time scale as a function of axial and radial location and Reynolds number with unprecedented spatial resolution. Multi-point time correlations are then used to examine the nature of scalar advection and spreading rate across the width of the jet. In addition to the statistical representation, both time scale and spreading rate are examined visually to gain an improved qualitative understanding of scalar mixing.

9:31AM G22.00008 How a scalar puff that is written in turbulence disperses: theory and experiment , WILLEM VAN DE WATER, Physics Department Eindhoven University of Technology, ENRICO CALZAVARINI, Laboratoire de Mécanique de Lille CNRS/UMR 8107, Université Lille, MEHRNOOSH MIRZAEI, Institute for Molecules and Materials, Radboud University of Nijmegen, BRUNO ECKHARDT, Fachbereich Physik, Philipps-Universität Marburg, FEDERICO TOSCHI, Physics Department, Eindhoven University of Technology, NICO DAM, Mechanical Engineering Department, Eindhoven University of Technology — When a blob of passive scalar is released in turbulence, it will spread due to the combined action of turbulence and molecular diffusion. It is a still unresolved question whether molecular diffusion helps or suppresses the spreading of the blob. We write a scalar puff in a strongly turbulent flow of air using molecular tagging with two crossed UV laser beams. The puff is made by fusing N_2 and O_2 molecules to NO, which is then used as a tracer. The dispersion of the puff is followed using laser–induced fluorescence. When the blob is small (size $\approx 10\,\eta$), the evolution of its Gaussian parameters Γ satisfies a simple linearized equation driven by the gradients of the turbulent velocity field [1]. It was computed using the velocity field of a direct numerical simulation ($\mathrm{Re}_\lambda = 400$). At short times we find striking agreement between experiment and numerical simulation. A question is whether the strongly anomalous statistics of the gradients will endow the fluctuations of Γ with special properties. A preliminary conclusion is that this is not the case, with the fluctuations being close to log–normal. [1] H. Tennekes and J. L. Lumley, *A First Course in Turbulence*.

9:44AM G22.00009 On the interaction of two scalar plumes in a turbulent flow , BING-CHEN WANG, SHAHIN OSKOUIE, Univ. of Manitoba, EUGENE YEE, Defence R&D Canada-Suffield — Direct numerical simulation is used to study the interaction of two plumes released by two point sources in the context of a turbulent open channel flow. This study is inspired by the classical experiment in quantum physics, Young's double-slit interference experiment on light and energy. The results of the first-order concentration statistics show that in the convective range, the two ground plumes mix faster in the spanwise direction, however, in the turbulent diffusion range, the elevated plumes spread and mix faster. It is observed that streamwise evolution of the second-order correlation function at the midpoint between the two plumes exhibits four distinct mixing stages for both ground and elevated sources. The second-order correlation function demonstrates that the degree of mixing is minimum at the midpoint between the two plumes and is maximum at plume fringes. In general, the elevated plumes exhibit higher degree of mixing in comparison with the ground plumes due to the meandering effects. The scatterplots prove that the higher order statistics of the concentration can be predicted by the knowledge of the first and second-order statistics.

Monday, November 25, 2013 8:00AM - 10:10AM –

Session G23 Turbulence: Theory IV - Modeling and Simulation 318 - William K. George, Princeton University

8:00AM G23.00001 On Intense Vortex Structures in Isotropic Turbulence , ANTHONY LEONARD, California Institute of Technology — We continue our study of vortex structures are 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). First, a connection is made between the PDF of a single intense structure and the vorticity distribution within that structure. Second, this PDF for intense structures, coupled with the distribution of finite-time Lyapunov exponents for material deformation in isotropic turbulence, yields a candidate for the full PDF of vorticity amplitudes.

8:13AM G23.00002 Self-sustaining turbulence in a Restricted Nonlinear (RNL) Model of plane Couette flow , VAUGHAN THOMAS, DENNICE GAYME, Johns Hopkins University, BRIAN FARRELL, Harvard University, PETROS IOANNOU, University of Athens — In this work we develop a restricted non-linear model (RNL) of plane Couette flow based on stochastic structural stability theory (S3T). The S3T system consists of a coupled set of equations for the evolution of a streamwise averaged mean flow forced by the ensemble averaged Reynolds stresses computed from the perturbations. The RNL model calculates the evolution of a single member of the perturbation ensemble interacting with the time varying streamwise averaged mean flow. Simulations of the RNL exhibit self-sustaining turbulent behavior that closely resembles DNS. S3T based analysis of this system shows that this self-sustaining activity arises due to the coupling from the mean flow to the perturbations, in other words the fact that the perturbations depend parametrically on the current state of the streamwise averaged mean flow. Elimination of this interaction reduces the system to the so-called 2D/3C model, which is asymptotically stable and consequently, does not exhibit turbulent behavior in the absence of forcing. Studies of the RNL confirm that the turbulence intensity decreases as the coupling strength is reduced, and that its behavior collapses to that of the 2D/3C model at a non-zero threshold.

8:26AM G23.00003 A minimal representation of turbulence in plane Couette flow , DENNICE F. GAYME, VAUGHAN THOMAS, Johns Hopkins University, BRIAN FARRELL, Harvard University, PETROS IOANNOU, University of Athens — We describe a stochastic structural stability theory (S3T) based model of fully developed turbulence in plane Couette flow. This model is obtained by partitioning Navier Stokes into a nonlinear equation governing the evolution of the streamwise averaged mean flow and a linearized equation for the covariance of streamwise varying perturbations. When coupled, these equations explicitly model the dynamics of a second order approximation of the probability distribution of the turbulence. We investigate this system using a computationally tractable Restricted Nonlinear (RNL) model that represents the dynamics of a single member of the infinite ensemble of the S3T system. The RNL system has been shown to capture the dynamics of roll/streak structures and to support self-sustaining turbulence. Our results demonstrate that this self-sustaining state naturally collapses to a minimal realization of turbulence that retains only the essential set of streamwise varying perturbations. Comparisons to DNS data show that this minimal representation captures the salient features of fully developed turbulence and that the wavelengths involved in this behavior are independent of the number of streamwise modes used or the channel length.

8:39AM G23.00004 A "resonant" spanwise perturbation frequency in streamwise-constant Couette flow , ISMAIL HAMEDUDDIN, DENNICE GAYME, The Johns Hopkins University — Turbulence in plane Couette flow is dominated by streamwise elongated structures that are approximately spanwise periodic with a preferred spatial frequency. It has been postulated that these approximately streamwise-constant coherent structures develop due to streamwise vortices in the flow. We investigate this idea by considering a streamwise-constant (2D/3C) model of plane Couette flow. We introduce streamwise vortices by imposing spanwise periodic cross-stream perturbations on the flow field and study it's energy amplification under stochastic disturbances. The periodic nature of the resulting equations allows us to cast the system into a convenient, so-called "lifted," form that retains the periodic coefficients in the analysis. We can then efficiently solve for the energy amplification using a perturbation approach on the associated Lyapunov equation. Our results show the existence of a peak or "resonant" spanwise frequency that maximizes the disturbance amplification, suggesting that the 2D/3C equations capture the type of (spanwise frequency) selective mechanism that leads to spanwise periodic structures common in fully developed flows.

8:52AM G23.00005 Exact coherent structures in 2D weakly turbulent flow , ROMAN GRIGORIEV, RAVI PALLANTLA, Georgia Institute of Technology — The description of fluid dynamics in terms of exact coherent structures (ECS) has recently emerged as a promising approach to a deterministic description of weak turbulence. Each ECS corresponds to an exact regular unstable solution of the Navier-Stokes equation and turbulence can be thought of as a walk through neighborhoods of a set of ECS. Although many ECS of different types have been identified numerically for a variety of experimentally realizable 3D flows (e.g., pipe Pouseuille and plane Couette flows), none have been verified to exist in experiment, in part due to the practical difficulties with setting up the appropriate initial conditions. In this talk we discuss numerically computed ECS in a model of a 2D Lorentz force-driven flow in a thin layer of electrolyte, which should be much easier to compare with experiment due to the relative ease with which 2D flow can be manipulation and observed. Special attention is given to enforcing physical boundary conditions and the choice of protocols that can be used in experiment to reproduce unstable flows corresponding to computed ECS.

9:05AM G23.00006 Universal Realizable Anisotropic Prestress (URAPS) Closure for the Reynolds Stress , CHARLES PETTY, Michigan State University, KARUNA KOPPULA, Rochester Institute of Technology, ANDRE BENARD, Michigan State University, MSU COLLABORATION — The Reynolds-averaged Navier-Stokes (RANS-) equation for constant property Newtonian fluids is unclosed due to the explicit appearance of the normalized Reynolds (NR-) stress and the turbulent kinetic energy. Clearly, any solution to an NS-closure model must be a non-negative operator. This longstanding problem has recently been addressed by developing a non-negative algebraic mapping of the NR-stress into itself. Consequently, all solutions of the URAPS NR-stress equation are non-negative dyadic-valued linear operators regardless of the class of benchmark flows used to determine closure parameters. Most significantly, unlike the class of Boussinesq closures for the NR-stress, the new theory predicts the redistribution of the turbulent kinetic energy among the three components of the fluctuating velocity field for statistically stationary spanwise rotating channel flows. Furthermore, the URAPS theory also predicts that the Coriolis acceleration causes an anisotropic re-distribution of turbulent kinetic energy among the three components of the fluctuating velocity field in rotating homogeneous decay.

9:18AM G23.00007 Simultaneous large-scale and sub-grid scale PIV measurements in a turbulent shear flow , OLIVER BUXTON, Imperial College London, BHARATHRAM GANAPATHISUBRAMANI, University of Southampton — An experimental investigation is undertaken in which the self-similar region of a nominally two-dimensional planar mixing layer is observed at inertial range and dissipative range spatial resolutions simultaneously. This is achieved by performing PIV experiments in which the field of view of three cameras, with a high spatial resolution, overlaps that of another camera with a lower spatial resolution in the far field of a mixing layer in which the Reynolds number based on the Taylor micro-scale is 260. The low-resolution experiment is thus analogous to a large eddy simulation (LES), in which the finest (sub-grid scale) stresses are modelled. This data thus permits an investigation of the effect of the large-scale fluctuations on the sub-grid scale (SGS) stresses, and vice-versa. It is found that the sign of the large-scale fluctuations is significant in determining the sub-grid scale activity, with low momentum (negative) large-scale fluctuations leading to an increase in the sub-grid scale stresses, particularly the $u'v'$ component. A Smagorinsky type SGS model is also compared to the experimental data in order to determine the effects of the large-scale fluctuations on the eddy viscosity.

9:31AM G23.00008 Turbulence in Taylor-Couette Flow and a Molecule Dependent Transport Equation , LUIS MA. BO-OT, National Institute of Physics, Univ of the Philippines Diliman QC Philippines, LUDEK JIRKOVSKY, Department of Informatics and Geo-informatics, Fakulta Zivotniho Prostredi, University of J.E. Purkyne, Usti n. L., Czech Republic — We apply a previously derived and utilized a modified Navier-Stokes equation to Taylor-Couette flow, that is fluid flow enclosed between two concentric cylinders where the inner cylinder is rotating with some constant speed and the outer cylinder is stationary or vice versa. We report first analytic solutions describing velocity profiles of such flow in turbulent regime. The analytic profiles are compared with results of the reported first direct numerical simulation of Taylor-Couette flow in turbulent regime [D. Pirro and M. Quadricio, Euro. J. of Mech. B, 27, (2008) 552-566]. PACS: 47.20.Qr, 47.27.-i, 02.30.Gp

9:44AM G23.00009 Temporal decorrelations in compressible isotropic turbulence , GUOWEI HE, XING ZHANG, DONG LI, LNM, Institute of Mechanics, Chinese Academy of Science — Temporal decorrelations in compressible isotropic turbulence are studied using the space-time correlation theory and direct numerical simulation. A swept-wave model is developed for dilatational components while the classic random sweeping model is proposed for solenoidal components. The swept-wave model shows that the temporal decorrelations in dilatational fluctuations are dominated by two physical processes: random sweeping and wave propagation. These models are supported by the direct numerical simulation of compressible isotropic turbulence, in the sense of that all curves of normalized time correlations for different wavenumbers collapse into a single one using the normalized time separations.

9:57AM G23.00010 ABSTRACT WITHDRAWN –

Monday, November 25, 2013 8:00AM - 9:57AM –

Session G24 Aerodynamics II 319 - Fulvia Scarano, TU Delft

8:00AM G24.00001 Dynamic instability of small-scale wind turbine blades[1] , PARIYA POURAZARM, YAHYA MODARRES-SADEGHI, MATTHEW LACKNER, University of Massachusetts Amherst — Future wind turbine blades will become larger, and therefore more flexible. For more flexible blades, the ratio of the estimated critical speed for dynamic instability to the operating speed decreases, and the blades are more susceptible to such instabilities. In the current work, the dynamic instability of a rotating wind turbine blade is studied using a numerical stability analysis and supported by experimental results. For the experimental component of the work, a series of tests were conducted in a wind tunnel. The blades were designed using relatively thin, low Reynolds number airfoils and built using rapid-prototyping methods with a flexible material. As the oncoming wind speed was increased, the beam natural frequencies varied, up to a critical wind speed at which two structural modes coalesced and resulted in a coupled-mode flutter. A theoretical model based on coupled flexural-torsional beam equations subjected to aerodynamic loadings is derived to study the flow-induced instability for the designed blade. The model also predicts the onset of instability at a critical wind speed at which one of the structural modes experiences a negative damping.

[1]The support provided by the Wind Technology Testing Center, a part of the Massachusetts Clean Energy Center is acknowledged.

8:13AM G24.00002 Studies of Mini-Turbines , STACEY CHAN, MASAKI ENDO, MICHAEL ROMANKO, C.H.K. WILLIAMSON, Fluid Dynamics Research Laboratories, Cornell University — Urban environments are inaccessible to large wind turbines of the classical "windmill" design. By exploring small-scale vertical-axis wind turbines (VAWTs), wind energy can possibly be harvested from the constrained spaces within cities. We present a comprehensive study of blade offset pitch angle and relative blade size (ratio of blade chordlength/turbine diameter, c/D). We find that the optimal pitch angle for a symmetric blade is the angle at which the midpoint chordline is tangent to the turbine circumference. Also, a turbine with conventional blades of small c/D ratio (c/D = 0.12) – typical of large scale turbines – do not operate well at low Reynolds numbers. On the other hand, the maximum coefficient of power for turbines with larger c/D ratio (c/D = 0.36) is much higher than for the conventional small-blades. As blade size increases, the operating range of TSR (Tip Speed Ratio) also increases, making large-chord turbines more robust to the prevailing wind conditions. Surprisingly, the regime of TSR for maximum power extracted, at these low Reynolds numbers, corresponds with small or even negative power predictions, based on streamtube theory.

8:26AM G24.00003 Performance Optimization and Analysis of Variable-Pitch Vertical-Axis Wind Turbines , DIETMAR REMPFER, PETER KOZAK, Illinois Institute of Technology — The blades of conventional vertical-axis wind turbines (VAWT) operate in a complex unsteady environment, characterized by periodically changing relative flow velocity and angle of attack, accentuated by passage through the wake of preceding blades. For many operating regimes, in particular for operation at low tip-speed ratio which is of interest in order to reduce mechanical loads, the blades experience dynamic stall, reducing overall efficiency and leading to significant torque fluctuations. Periodic pitch variation of the turbine blades may therefore be considered in order to avoid stall and increase efficiency. In this presentation we will discuss gains in operating characteristics and efficiency that can be obtained by such a strategy. We will describe a full optimization of turbine efficiency based on double-multiple streamtube models. In addition, we will compare these results, and discuss the physics of the associated flows using data obtained from two-dimensional Navier-Stokes simulations of such turbines. It will be shown that, while peak efficiency of a variable-pitch VAWT is only moderately higher than the one of a conventional fixed-pitch VAWT, we can achieve a much broader maximum, leading to significantly improved performance in practical use.

8:39AM G24.00004 Fluid-Structure Interaction Simulations of a Parked Wind Turbine Rotor Blade under Steady and Unsteady Inflow Conditions , ROBERT CAMPBELL, BALAJI JAYARAMAN, ADAM LAVELY, JAVIER MOTTA-MENA, GANESH VIJAYAKUMAR, Penn State University — Tightly coupled fluid-structure interaction (FSI) simulations are performed for an NREL 5 MW rotor blade in the parked configuration for steady and unsteady inflow conditions. The FSI solver employs a partitioned approach that couples OpenFOAM as the flow solver and an author-developed structural finite element solver. Sub-iterations are employed to ensure convergence of the flow and structural response every solution time step. The simulations are performed for the NREL 5 MW blade, with the structural response represented by a modal summation solution. A custom fluid mesh motion solver allows the fluid mesh motion to occur primarily in a region local to the blade, while maintaining the mesh quality near the blade surface. The time-accurate blade response allows the approximation of a linear structural model to be assessed for the NREL 5 MW blade. Details of the FSI solver, including the mesh motion scheme and solution times are presented. Comparisons of blade loadings for steady and unsteady inflow conditions demonstrate the importance of blade flexibility for these simulations. Supported by the US Department of Energy.

8:52AM G24.00005 Proper Orthogonal Decomposition analysis of Large Eddy Simulation data of a single wind turbine wake with uniform inflow[1] , CLAIRE VERHULST, Johns Hopkins University, ROBERT MIKKELSEN, JENS NORKAER SORENSEN, Technical University of Denmark, CHARLES MENEVEAU, Johns Hopkins University — Large Eddy Simulations have been performed using the EllipSys3D code to model the NREL 5 MW reference wind turbine with uniform incoming flow at 10 m/s. Instantaneous snapshots of the velocity field are decomposed using a fully three-dimensional Proper Orthogonal Decomposition (POD) analysis. This method unambiguously identifies the bases of velocity fields that most efficiently represent the turbulent kinetic energy of the snapshots on average. The structure of the resulting POD modes and the evolution of their magnitude in time provide insight into the dynamics of breakdown of tip vortices and recovery of the wake velocity deficit. In this presentation, we will discuss the structure of the first few POD modes and how the observed structure relates to the dynamics of the wind turbine wake.

[1]This research is supported by a NSF Graduate Fellowship and by the WINDINSPIRE project, funded through NSF-OISE 1243482

9:05AM G24.00006 Evaluation of drag forcing models for vertical axis wind turbine farms[1] , BRIAN PIERCE, PARVIZ MOIN, Stanford University, JOHN DABIRI, California Institute of Technology — Vertical axis wind turbines (VAWTs) have the potential to produce more power per unit area than horizontal axis wind turbines (HAWTs) in a wind farm setting (Kinzel et al. J. Turb. [2012]), but further understanding of the flow physics is required to design such farms. In this study we will model a large wind farm of VAWTs as an array of 100 circular cylinders which will allow a comparison with a laboratory experiment (Craig et al. DFD 2013). The geometric complexity and high Reynolds numbers necessitate phenomenological modeling of the interaction of the turbine with the fluid, which is done through point drag models similar to those found in canopy flow simulations (e.g. Dupont et al. J. Fluid Mech. [2010]). We will present a detailed study of the point drag model performance for flow over one cylinder, providing an evaluation of the model's fidelity as it relates to quantities of interest for the VAWT farm. Next we will present results for flow through the cylinder array, emphasizing validation of the model and insight into VAWT wind farm dynamics. We will also discuss the effect of wall modeling on the calculations, as the Reynolds number of the problem requires the application of wall modeling of the turbulent boundary layer above the ground to keep the cost manageable.

[1]Brian Pierce acknowledges support from the Stanford Graduate Fellowship

9:18AM G24.00007 Large eddy simulations of vertical axis wind turbines to optimize farm design , SEYED HOSSEIN HEZAVEH, ELIE BOU-ZEID, Princeton University — Wind energy production, and research have expanded considerably in the past decade. These efforts aim to reduce dependence on fossil fuels and the greenhouse gas emissions associated with current modes of energy production. However, with expanding wind farms, the land areas occupied by such farms become a limitation. Recently, interest in vertical axis wind turbines (VAWTs) has increased due to key advantages of this technology: compared to horizontal axis turbines, VAWTs can be built with larger scales, their performance is not sensitive to wind direction, and the ability to place their generators at the bottom of the mast can make them more stable offshore. In this study, we focus on how the Atmospheric Boundary Layer (ABL) will react to the presence of large VAWT farms. We present a state-of-art representation of VAWTs using an actuator line model in a Large Eddy Simulations code for the ABL. Validations are made against several experimental datasets, which include flow details and power coefficient curves, the wake of an individual turbine is visualized and analyzed, and the interaction of adjacent turbines is investigated in view of optimizing their interactions and the configuration of VAWT farms.

9:31AM G24.00008 Numerical study of ocean wave effect on offshore wind farm[1] , LIAN SHEN, Department of Mechanical Engineering and St. Anthony Falls Laboratory, University of Minnesota, DI YANG, CHARLES MENEVEAU, Department of Mechanical Engineering, Johns Hopkins University — Wind power at sea has become increasingly important in renewable energy study. For energy harvesting, winds over oceans have many advantages over winds on land, for example, larger and open surface area, faster wind speed, and more wind resource close to high population regions. On the other hand, the presence of ocean waves introduces complexities to wind turbines. There is a critical need to study the dynamical interactions among marine atmospheric boundary layer, ocean wave field, and floating turbines. In this research, we study offshore wind farm by performing large-eddy simulations for winds coupled with potential-flow-theory based simulations for broadband irregular waves, with the wind turbines represented by an actuator disk model. Our results show that windseas at different development stages result in different sea-surface roughness and have an appreciable effect on wind profile and the energy extraction rate of the turbines. If swells are present, swell-to-wind momentum and energy transfer further changes the wind field to introduce oscillations in as well as modify the mean of the wind power.

[1]DY and LS acknowledge the support of NSF-CBET-1341062. CM acknowledges the support of NSF-AGS-1045189 and NSF-OISE-1243482.

9:44AM G24.00009 Power Optimization of Wind Farms in Large Eddy Simulations[1] , JOHAN MEYERS, JAY PRAKASH GOIT, Mechanical Engineering, KU Leuven, Celestijnenlaan 300A, B3001 Leuven — As the understanding of wind-farm aerodynamics broadens, our interest is shifting towards exploring the possibilities of optimising and improving the power-extraction of a wind farm. In the present work we couple flow simulations performed using Large Eddy Simulations (LES) with gradient based optimization to control individual turbine in a farm, so as to achive an increase in the total power. The controls in our optimization problem are the thrust coefficients $C'_{\mathrm{T,n}}(t)$ of individual turbines as function of time. We use a gradient-based algorithm for the optimization and the gradients are computed using the adjoint method. In the first step we verify the adjoint calculated gradient by comparing it to the forward simulation based gradient obtained from finite difference of the cost function and find that errors remain below 5%. We further elaborate the optimization techniques, and present results for a number of cases of wind-farm boundary layer cases. We also discuss how the thrust coefficient $C'_{\mathrm{T,n}}(t)$ evolves with time for different turbine locations. We also present and interpret results of the adjoint fields.

[1]J.M. acknowledges support from the European Research Council (FP7-Ideas, grant no. 306471)

Monday, November 25, 2013 8:00AM - 10:10AM –

Session G25 Flow Control IV: Plasma Actuators 320 - Noel Clemens, University of Texas at Austin

8:00AM G25.00001 Real-time control of the boundary layer disturbance induced by a dynamic isolated roughness element using plasma actuators[1] , KYLE BADE, AHMED NAGUIB, Michigan State University, RONALD HANSON, PHILIPPE LAVOIE, University of Toronto Institute for Aerospace Studies, BRANDT BELSON, CLARENCE ROWLEY, Princeton University — It is well established that bypass boundary layer transition is initiated by the formation and growth of unsteady streaks. Motivated by the delay/prevention of transition, this study examines the ability to sense unsteady streaks in a Blasius boundary layer and to attenuate their transient growth. The unsteady streaks are introduced into the boundary layer using an isolated roughness element that is dynamically actuated from flush with the wall to a specified height; resulting in a time varying disturbance. A real-time, closed-loop, feedforward-feedback control system is designed to apply an appropriate voltage to a plasma actuator in order to reduce the roughness induced disturbance. The control system inputs come from two in-wall hot-wire shear stress sensors located within a high-speed streak disturbance, one upstream and one downstream of the plasma actuator. The controller is shown to effectively drive the shear stress at the feedback sensor toward the Blasius level. The flow state is later examined over a cross-flow plane above the feedback sensor to assess the effectiveness of the control in reducing the total disturbance energy. In addition, the effects of the control parameters on the controller's effectiveness and robustness are investigated.

[1]NSF Grant: CMMI 0932546

8:13AM G25.00002 Transition delay by introducing spanwise velocity gradients[1] , B.E.G. FALLENIUS, Linné Flow Centre, KTH Mechanics, K. BARCKMANN, Center of smart interfaces, TU Darmstadt, J.H.M. FRANSSON, Linné Flow Centre, KTH Mechanics, S. GRUNDMANN, Linné Flow Centre, KTH Mechanics, Center of smart interfaces, TU Darmstadt — For stabilizing a boundary layer it has been shown both numerically and experimentally that the control idea of introducing steady spanwise velocity gradients[2] is far more effective than what could be foreseen. Different devices have been analyzed experimentally, which can modulate the boundary layer in a controlled way, and so far miniature vortex generators[3] have shown to be the most coveted for transition delay. Currently, a popular control device in the control community is the plasma actuator, which mainly has shown its potential for separation control, but lately also for transition delay. In a wind tunnel investigation at KTH, the plasma actuators have been tested in a configuration aimed at making use of above control idea and its potential for transition delay will be discussed.

[1]Support from the European Research Council is acknowledged.

[2]Cossu, C., Brandt, L. *Eur. J. Mech./B Fluids* **23**, 815, 2004.

[3]Shahinfar, S., Sattarzadeh, S. S., Fransson, J. H. M., Talamelli, A. *Phys. Rev. Lett.* **109**, 074501, 2012.

8:26AM G25.00003 Effect of Pulsed Plasma Jets on Reflected Shock-Turbulent Boundary Layer Interaction[1] , BENTON R. GREENE, NOEL T. CLEMENS, UT Austin Dept. of Aerospace Engineering and Engineering Mechanics, PATRICK MAGARI, DANIEL MICKA, Creare, Inc. — Shock-induced turbulent boundary layer separation can have many detrimental effects in supersonic inlets including flow instability, fatigue of structural panels, poor pressure recovery, and unstart. Pulsed plasma jets (or "spark jets"), zero net mass flow jets characterized by high bandwidth and the ability to direct momentum into the flow, are one promising method of reducing shock-induced separation and boundary layer distortion. The current study is focused on investigating the efficacy of pulsed plasma jets to reduce the boundary layer distortion induced by a reflected shock interaction in a Mach 3 flow. A 7° shock generator placed outside the tunnel ceiling boundary layer produces an incident shock on the floor of the tunnel of sufficient strength to induce separation. An array of pulsed plasma jets are placed approximately 2 boundary layer thicknesses upstream of the interaction and pulsed at between 1 kHz and 4 kHz. PIV is used to investigate the effect of the jets on the nature of the separation as well as the boundary layer distortion and pressure recovery downstream of the interaction.

[1]Funded through AFRL in collaboration with Creare, Inc.

8:39AM G25.00004 The Influence of Spanwise Segmented Plasma Actuator Forcing on a Circular Cylinder Wake and the Selection of Optimum Wavelength[1] , SAMIK BHATTACHARYA, JAMES W. GREGORY, The Ohio State University — Detailed investigations have been carried out on the effect of segmented plasma forcing on the wake of circular cylinder. Actuators of wavelength 1d to 6d (d=diameter) were used for three-dimensional actuation at Reynolds number of 4700. Two most important factors were wavelength of actuation and the power of plasma. Vortex shedding was not significantly attenuated below a certain threshold of the supplied voltage. However, for actuation wavelength more than 2d, the near wake developed a wavy profile due to emergence of streamwise vorticity. The reason for this development was differential displacement of the Karman vortex street behind the plasma forming and no plasma region. Forcing above the threshold voltage created strong circulating zones at each corner of the buried electrode, which diverted the flow from the no plasma region towards the plasma region. This process gave rise to alternate accelerated flow (behind the plasma region) and distinct reverse flow zones (behind no plasma regions). The strength of Karman shedding was attenuated as energy was extracted from it and fed to streamwise vorticity. This lead to significant reduction in drag in the event of high power actuation with 3d,4d and 5d actuators. The attenuation in shedding and reduction of drag was maximum for 4d actuator, which led to its selection as optimum wavelength. Significant difference in wake width was observed in high power forcing cases behind the two regions. This observation was attributed to spanwise difference in vortex formation length due to segmented forcing.

[1]This work was supported by the Air Force Office of Scientific Research.

8:52AM G25.00005 Experimental Study of the Power Profile Airfoil Equipped with Plasma Flow Control , LIBIN DANIEL, JAMEY JACOB, Oklahoma State University — This presentation discusses results from an experimental study of the power profile airfoil at low Reynolds number. The power profile airfoil was developed by AMO Smith and consists of a blunt trailing edge shape with two wall jets near the trailing edge. The replacement of streamlining with properly designed blowing is used to prevent flow separation and additionally offers potential applications as a powered high-lift system, propulsive system, or low inertia control device. The 2D wind-tunnel model consists of the 22.5% thick power profile airfoil equipped with a movable trailing edge plug to direct flow along the trailing edge streamline. Compressed air was passed into the model via a plenum with flow conditioning devices to create pressure backdrop to allow uniform blowing at the trailing edge. The effects of varying jet momentum coefficient and trailing edge positioning on the aerodynamic characteristics are observed with both wake surveys and PIV. The impact of plasma synthetic jet actuators (PSJA) placed along the trailing edge of the power profile airfoil is also discussed. PSJA operation is compared to the baseline power profile airfoil both alone and working with the blowing to provide additional control authority.

9:05AM G25.00006 Vane Separation Control in a Linear Cascade with Area Expansion using AC DBD Plasma Actuators , CHRISTOPHER KLEVEN, THOMAS CORKE, University of Notre Dame — Experiments are presented on the use of AC dielectric barrier discharge (DBD) plasma actuators to prevent flow separation on vanes in a linear cascade with area expansion. The inlet Mach number to the cascade ranged from 0.3 to 0.5, and the vane chord Reynolds numbers ranged from 0.9×10^6 to 1.5×10^6. Three cascade designs with different amounts of area expansion, providing different degrees of adverse pressure gradients, were examined. Surface flow visualization revealed a 3-D separation bubble with strong recirculation that formed on the suction side of the vanes. The pattern agreed well with CFD simulations. Plasma actuators were placed on the suction sides of the vanes, just upstream of the flow separation location. Quantitative measurements were performed in the wakes of the vanes using a 5-hole Pitot probe. The measurements were used to determine the effect of the plasma actuator separation control on the pressure loss coefficient, and flow turning angle through the cascades. Overall, the plasma actuators separation control increased the velocity magnitude and dynamic pressure in the passage between the vanes, resulted in a more spanwise-uniform flow turning angle in the vane passage, and significantly lowered the loss coefficient compared to the baseline.

9:18AM G25.00007 An Investigation of Plasma Actuators for Flow Control in a 140° Bend , MICHAEL ARTHUR, THOMAS CORKE, THOMAS SAMPER, University of Notre Dame — An experiment is presented to examine the effectiveness of AC dielectric barrier discharge (DBD) plasma actuators to maintain attached flow around a 140° bend in a sector of a azimuthal channel. The Mach number at the inlet to the bend ranges from 0.10 to 0.23, and the static pressure ranges from 7 to 22 atmospheres. The plasma actuator is located just upstream of the natural flow separation location. It is designed to produce a net body force that is directed downstream and towards the wall. The requirements and design of the plasma actuator were augmented by a CFD simulation that included the plasma actuator body force vector field. The measurements consist of the static pressure distribution along the outside wall of the bend, and an array of total pressure sensors across the centerline of the exit of the bend. The pressure distribution across the channel showed the desired flattening of the mean pressure profile with increasing AC voltage and frequency that was indicative of a more uniform mass flow profile at the exit of the bend. These confirmed a plasma actuator effect that scaled linearly with the AC frequency, and with AC voltage to a power that is expected from the AC DBD analytical model.

9:31AM G25.00008 DBD Control of a Turbulent Shear Layer downstream of a Backward Facing Step[1] , PATRICIA SUJAR-GARRIDO[2], NICOLAS BENARD[3], ERIC MOREAU[4], JEAN-PAUL BONNET[5], Institut Pprime CNRS University of Poitiers ENSMA — The present paper deals with the control of a free shear layer downstream of a backward-facing-step of height 3 cm at Re 3x10^4. The initial boundary layer thickness is 1.2 cm with Re$_\theta$ 1200. The control is achieved via a single Dielectric Barrier Discharge (DBD). An optimal frequency is observed and it is shown that the plasma discharge is able to manipulate the first stages of the formation of the free shear layer and consequently to modify the flow dynamics of the entire flow; the results show some limitations of the control authority of this type of plasma discharge. Time unresolved and resolved measurements techniques are used to investigate the influence of plasma device on the global modification of mean reattachment length and on the large-scale structures and turbulent energy distribution.

[1]This work was supported by the 7th Framework Program FP7/2010-2013 MARS
[2]PhD student
[3]Assistant Professor
[4]Professor
[5]Research Director, CNRS

9:44AM G25.00009 Active Control of Natural Tollmien-Schlichting Waves using Plasma Actuators , MARIOS KOTSONIS, RAM KRISHAN SHUKLA, STEFAN PROBSTING, Delft University of Technology — An experimental study is performed on active control of boundary layer instabilities developing on a NACA 0012 airfoil. A closed-loop control system has been implemented using the filtered-x Least Mean Squares adaptive algorithm based on Finite Impulse Response filters. Surface mounted microphones are used as sensors. The controller drives a Dielectric Barrier Discharge plasma actuator placed along the span of the airfoil. In contrast to the conventional sinusoidal signal, the actuator is powered using a continuously adapted signal selected by the controller in order to damp the incoming wavetrain of TS waves. High speed 2-component Particle Image Velocimetry is used to characterize the flow in the vicinity of the actuator. Several cases are tested using both open-loop and closed-loop actuation. Tested freestream velocities range from 17 to 25 m/s at chord Reynolds of 0.22 to 0.33 million respectively. Results indicate the suppression of the tonal component of unstable TS waves with closed-loop actuation. Amplitude reduction of approximately 50 $\%$ is achieved for freestream velocity of 17 m/s while significant suppression is maintained for higher velocities. In the case of open-loop control, the actuator is operated using non-adapted single-frequency sinusoidal signal.

9:57AM G25.00010 Suppression or Enhancement of Pressure Fluctuations in High Subsonic Cavity Flow Using Plasma Actuators[1] , MO SAMIMY, KEVIN YUGULIS, SAMUEL HANSFORD, JAMES GREGORY, The Ohio State University, GDTL/AARL TEAM — Localized arc filament plasma actuators (LAFPAs) were used to control pressure fluctuations in a Mach 0.6 cavity flow with a Reynolds number based on the cavity depth of 200,000. The rear wall of the cavity is inclined 30° with respect to the upstream flow and the cavity depth and length-to-depth ratio are 12.7 mm and 4.86. Five actuators were uniformly distributed along the span of the cavity 1 mm upstream of the cavity leading edge. Forcing was conducted quasi-two-dimensionally (all actuators operated in phase) and three-dimensionally (actuators operated out of phase). Time-resolved pressure and PIV measurements were used to assess the effectiveness of the actuators and to explore the physics of the flow. The results show that with proper selection of forcing frequency; (1) the cavity tone can be suppressed by over 20 dB and the broadband pressure fluctuations can be suppressed by over 5 dB in a strongly resonating cavity, and (2) the resonance can be re-established and the peak tone as well as the broadband pressure fluctuations can be significantly amplified in a weakly resonating cavity. Both quasi-two-dimensional and three-dimensional controls were successful in controlling the flow, but the control was less sensitive to forcing frequency change in the latter than former; and much wider forcing frequency range could be used in case (1) than case (2).

[1]Supported by AFRL/ISSI.

Monday, November 25, 2013 8:00AM - 10:10AM –

Session G26 Reacting Flows IV: PDF/FDF 321 - Reza H. Sheikhi, Northeastern University

8:00AM G26.00001 PDF modeling of chemically reacting flows in a compression-ignition engine , VIVEK RAJA RAJ MOHAN, DANIEL HAWORTH, Penn State University, JIAN LI, Volvo Group Trucks Technology — A transported probability density function (PDF) model is used to simulate the in-cylinder combustion processes in a compression-ignition heavy-duty engine. The flow inside the cylinder in a compression-ignition engine is chemically reacting and highly turbulent. Therefore, the turbulent fluctuations in composition and temperature will influence the mean reaction rates. These turbulence-chemistry interactions (TCI) play an important role in predicting the combustion processes accurately. Recent results from in-cylinder combustion simulations for a compression-ignition engine are compared with measured data for several operating conditions. The PDF model, which takes into account for TCI, predicts the combustion processes more accurately compared to a model which neglects TCI. Marked differences are observed in predicting the flame structure and the pressure and heat-release traces as well as in predicting the emission characteristics.

8:13AM G26.00002 A Partially-Stirred Batch Reactor Model for Under-Ventilated Fire Dynamics , RANDALL MCDERMOTT, CRAIG WEINSCHENK, National Institute of Standards and Technology, Gaithersburg, Maryland — A simple discrete quadrature method is developed for closure of the mean chemical source term in large-eddy simulations (LES) and implemented in the publicly available fire model, Fire Dynamics Simulator (FDS). The method is cast as a partially-stirred batch reactor model for each computational cell. The model has three distinct components: (1) a subgrid mixing environment, (2) a mixing model, and (3) a set of chemical rate laws. The subgrid probability density function (PDF) is described by a linear combination of Dirac delta functions with quadrature weights set to satisfy simple integral constraints for the computational cell. It is shown that under certain limiting assumptions, the present method reduces to the eddy dissipation concept (EDC). The model is used to predict carbon monoxide concentrations in direct numerical simulation (DNS) of a methane slot burner and in LES of an under-ventilated compartment fire.

8:26AM G26.00003 Subfilter Modeling in Spray Combustion Using the Probability Density Function Approach , COLIN HEYE, VENKAT RAMAN, The University of Texas at Austin — A probability density function (PDF) based approach for modeling spray combustion in the large eddy simulation (LES) context is used to study a series of experimental spray flames. Complex coupling of droplet dispersion, evaporation and scalar mixing in turbulent spray-laden flows results in a range of combustion regimes. Prior work has shown that variations in fuel inflow conditions can change the flame structure, however significant simplifications were made in these simulations with the use of steady laminar flamelet based models. In the joint-scalar PDF transport equation, the chemical source term appears closed, and in this work, in situ adaptive tabulation is effectively utilized to calculate component source terms allowing for the impact of finite rate kinetics to be analyzed. Further, the correlation between the evaporation source term and the subfilter scalar PDF is analyzed. Results from a priori direct numerical simulation (DNS) studies and LES calculations will be presented.

8:39AM G26.00004 Numerical simulations of turbulent jet ignition and combustion , ABDOULAHAD VALIDI, ABOLFAZL IRANNEJAD, FARHAD JABERI, Michigan State University — The ignition and combustion of a homogeneous lean hydrogen-air mixture by a turbulent jet flow of hot combustion products injected into a colder gas mixture are studied by a high fidelity numerical model. Turbulent jet ignition can be considered as an efficient method for starting and controlling the reaction in homogeneously charged combustion systems used in advanced internal combustion and gas turbine engines. In this work, we study in details the physics of turbulent jet ignition in a fundamental flow configuration. The flow and combustion are modeled with the hybrid large eddy simulation/filtered mass density function (LES/FMDF) approach, in which the filtered form the compressible Navier-Stokes equations are solved with a high-order finite difference scheme for the turbulent velocity and the FMDF transport equations are solved with a Lagrangian stochastic method to obtain the scalar (temperature and species mass fractions) field. The hydrogen oxidation is described by a detailed reaction mechanism with 37 elementary reactions and 9 species.

8:52AM G26.00005 LES-Based Analysis of Entropy Generation in a Turbulent Nonpremixed Flame , MEHDI SAFARI, REZA H. SHEIKHI, Northeastern University, NUCEC TEAM — Entropy generation analysis is an effective means of improving the efficiency of turbulent combustion from the second law of thermodynamics standpoint. Large eddy simulation (LES) of turbulent reacting flows is conducted with inclusion of entropy transport. The filtered form of this equation includes irreversible losses by entropy production due to viscous dissipation, heat conduction, mass diffusion and chemical reaction, all of which appear as unclosed terms. The closure is provided by a novel methodology entitled scalar, entropy filtered density function (SEn-FDF). The SEn-FDF describes the transport and generation of entropy, and is governed by an exact transport equation. This equation is modeled by a set of stochastic differential equations, which is solved by a Lagrangian Monte Carlo method. The main advantage of the SEn-FDF is that it provides closure for all individual entropy generation modes. It also includes the effects of chemical reaction in closed forms. The methodology is applied to a turbulent nonpremixed jet flame (Sandia Flame D) and predictions are assessed against experimental data. Entropy generation modes are obtained from the SEn-FDF and analyzed.

9:05AM G26.00006 FDF Simulation of the PRECCINSTA Burner , NASEEM ANSARI, University of Pittsburgh, ANSYS Inc., GRAHAM M. GOLDIN, ANSYS Inc., PETER A. STRAKEY, National Energy Technology Laboratory, PEYMAN GIVI, University of Pittsburgh — Since its original development over a decade ago, the filtered density function (FDF) has experienced widespread application for LES of a variety of turbulent reacting flows. The present work demonstrates that the FDF can now be considered for LES of complex flames in complex combustors. This is done by implementation of the scalar FDF on a domain portrayed by an unstructured grid. The modeled transport equation for the FDF is solved by a Lagrangian Monte Carlo method, coupled with the finite-volume solution of the transport flow variables. The resulting hybrid solver is employed for LES of the PRECCINSTA burner from DLR. The predictive capability of the FDF is assessed by comparison of the Reynolds-averaged statistics of the thermo-chemical variables with measured data. In general, the agreements are very good. This warrants future applications of the methodology for LES of practical combustors.

9:18AM G26.00007 Parametric modeling studies of turbulent non-premixed jet flames with thin reaction zones , HAIFENG WANG, Purdue University — The Sydney piloted jet flame series (Flames L, B, and M) feature thinner reaction zones and hence impose greater challenges to modeling than the Sanida Piloted jet flames (Flames D, E, and F). Recently, the Sydney flames received renewed interest due to these challenges. Several new modeling efforts have emerged. However, no systematic parametric modeling studies have been reported for the Sydney flames. A large set of modeling computations of the Sydney flames is presented here by using the coupled large eddy simulation (LES) /probability density function (PDF) method. Parametric studies are performed to gain insight into the model performance, its sensitivity and the effect of numerics.

9:31AM G26.00008 Large Eddy Simulation / Filtered Mass Density Function Modeling of High Pressure Turbulent Hydrogen Flames[1] , RICHARD MILLER, ZHIYUAN MA, Clemson University — The hybrid Large Eddy Simulation / Filtered Mass Density Function (LES/FMDF) approach to turbulent combustion simulations is extended to include high pressure physics. A posteriori simulations of an existing database of Direct Numerical Simulations (DNS) of high pressure turbulent hydrogen-oxygen and hydrogen-air flames are presented. The DNS include a real fluid equation of state, realistic pressure dependent property models, generalized heat and mass diffusion derived from non-equilibrium thermodynamics and fluctuation theory, and a detailed pressure dependent chemical kinetics mechanism. The geometry considered is a temporally developing reacting shear layer flame. The DNS are conducted at initial shear layer Reynolds numbers up to 4,500 and for pressures as large as 125 atm on numerical meshes up to approximately 3/4 billion grid points. Proper implementation of the LES/FDF approach to simulating the DNS flames is discussed and the simulation results are compared to the filtered DNS results.

[1]NSF Grant CBET-0965624

9:44AM G26.00009 A New LES/PDF Method for Computational Modeling of Turbulent Reacting Flows[1] , HASRET TURKERI, METIN MURADOGLU, Department of Mechanical Engineering, Koc University, Rumelifeneri Yolu, Sariyer 34450 Istanbul, Turkey, STEPHEN B. POPE, Sibley School of Mechanical and Aerospace Engineering, Cornell University, Ithaca, NY, 14853 USA — A new LES/PDF method is developed for computational modeling of turbulent reacting flows. The open source package, OpenFOAM, is adopted as the LES solver and combined with the particle-based Monte Carlo method to solve the LES/PDF model equations. The dynamic Smagorinsky model is employed to account for the subgrid-scale motions. The LES solver is first validated for the Sandia Flame D using a steady flamelet method in which the chemical compositions, density and temperature fields are parameterized by the mean mixture fraction and its variance. In this approach, the modeled transport equations for the mean mixture fraction and the square of the mixture fraction are solved and the variance is then computed from its definition. The results are found to be in a good agreement with the experimental data. Then the LES solver is combined with the particle-based Monte Carlo algorithm to form a complete solver for the LES/PDF model equations. The in situ adaptive tabulation (ISAT) algorithm is incorporated into the LES/PDF method for efficient implementation of detailed chemical kinetics. The LES/PDF method is also applied to the Sandia Flame D using the GRI-Mech 3.0 chemical mechanism and the results are compared with the experimental data and the earlier PDF simulations.

[1]The Scientific and Technical Research Council of Turkey (TUBITAK), Grant No. 111M067

9:57AM G26.00010 A New Hybrid FV/Particle Algorithm for PDF Simulations of Turbulent Reacting Flows[1] , REZA MOKHTARPOOR, MRTIN MURADOGLU, Koc University — A new consistent hybrid finite-volume (FV)/particle method is developed for solving the joint PDF equations of turbulent reacting flows. The open source FV package, OpenFOAM, is employed to solve the Favre-averaged mean mass and momentum equations while a particle-based Monte Carlo method is used to solve the fluctuating velocity-turbulence frequency-compositions JPDF transport equation. This work is motivated and designed to eliminate the deficiencies of the hybrid algorithm developed by Muradoglu et al. (1999, 2001). In the earlier hybrid method, a density-based FV algorithm was used to solve the mean flow equations, which has been found to be too dissipative for low-Mach number flows mainly due to the stiffness of the compressible flow equations in this limit. For tackling this problem, the density-based FV solver is replaced with a pressure-based PISO algorithm in the OpenFOAM package. The method is then applied to simulate the Sydney non-swirling and swirling bluff-body stabilized flames and the results are found to be in a good agreement with the experimental data and with the earlier PDF simulations of the same flames.

[1]The Scientific and Technical Research Council of Turkey (TUBITAK), Grant No. 112M181

Monday, November 25, 2013 8:00AM - 10:10AM –

Session G28 Waves II Spirit of Pittsburgh Ballroom B/C - Leonardo Chamorro, University of Illinois at Urbana-Champaign

8:00AM G28.00001 Scaling and kinematics of a floating wind turbine under ocean waves and variable thrust: an experimental study , CHRIS FEIST, St. Anthony Falls Lab, UMN, KELLEY RUEHL, Water Power Technologies, Sandia National Laboratories, MICHELE GUALA, St. Anthony Falls Laboratory, UMN — Scale model wave channel experiments were performed to study the motion of an offshore floating wind turbine in operational sea states. The model tests were conducted on a 1:100 Froude scaled Sandia National Labs 13.2 MW prototype offshore wind turbine with a barge style floating platform. The platform is modeled after the MIT/NREL Shallow Drafted Barge designed for the 5MW Offshore Baseline wind turbine. The wave environment used in the model tests is representative of the deep-water sea states off the coast of Maine as well as the Pacific Northwest. The purpose of the tests is to validate a computational model of the turbine-wave interaction where the effects of airflow are not considered. To simplify the tests and validation, the platform motion is restricted to two degrees of freedom, pitch and heave, by attaching two roller support types at the center of gravity along the pitch axis. The major aerodynamic force acting on the turbine, i.e. the rotor thrust, is provided by spinning a scaled rotor at a controlled rotational speed. A subset of experiments were performed to study the effect of a mean or fluctuating rotor thrust on the platform dynamics, exploring strategies to control the thrust as a function of platform pitch angle and minimize platform oscillations.

8:13AM G28.00002 Fluid-structure interaction simulation of floating wind turbines interacting with complex, large-scale ocean waves[1] , ANTONI CALDERER, XIN GUO, LIAN SHEN, FOTIS SOTIROPOULOS, St. Anthony Falls Lab., University of Minnesota — We develop a numerical method for simulating coupled interactions of complex floating structures with large-scale ocean waves and atmospheric turbulence. The Fluid-Structure Interaction (FSI) solver integrates the curvilinear immersed boundary method of Borazjani et al. (JCP 2008) with the level-set method of Kang et al. (Adv. in Water Res. 2012) and is capable of simulating the coupled dynamic interaction of arbitrarily complex bodies with airflow and waves. The large-scale wave model is based on the two-fluid coupled approach of Yang et al. (JCP 2011), which employs a high-order spectral method for simulating the water motion and a viscous solver with undulatory boundaries for the air motion. The large-scale wave field solver is coupled with the near-field FSI solver by feeding into the latter large-scale waves via the pressure-forcing method of Guo et al. (JCP 2009), appropriately adapted herein for the level set method. We validate the model under both simple wave trains and three-dimensional directional waves and compare the results with experimental and theoretical solutions. Finally, we demonstrate the capabilities of the new solver by carrying out large eddy simulation of a floating offshore wind turbine platform interacting with realistic ocean waves.

[1]This work is supported by the US Department of Energy (DE-EE0005482), the National Science Foundation (CBET-1341062), the University of Minnesota Initiative for Renewable Energy and the Environment, and the Minnesota Supercomputing Institute.

8:26AM G28.00003 Impact of plunging breaking waves on a partially submerged cube[1] , A. WANG, C. IKEDA, J.H. DUNCAN, University of Maryland, College Park — The impact of a deep-water plunging breaking wave on a partially submerged cube is studied experimentally in a tank that is 14.8 m long and 1.2 m wide with a water depth of 0.91 m. The breakers are created from dispersively focused wave packets generated by a programmable wave maker. The water surface profile in the vertical center plane of the cube is measured using a cinematic laser-induced fluorescence technique with movie frame rates ranging from 300 to 4,500 Hz. The pressure distribution on the front face of the cube is measured with 24 fast-response sensors simultaneously with the wave profile measurements. The cube is positioned vertically at three heights relative to the mean water level and horizontally at a distance from the wave maker where a strong vertical water jet is formed. The portion of the water surface between the contact point on the front face of the cube and the wave crest is fitted with a circular arc and the radius and vertical position of the fitted circle is tracked during the impact. The vertical acceleration of the contact point reaches more than 50 times the acceleration of gravity and the pressure distribution just below the free surface shows a localized high-pressure region with a very high vertical pressure gradient.

[1]This work is supported by the Office of Naval Research under grant N000141110095.

8:39AM G28.00004 Slamming pressures on the bottom of a free-falling vertical wedge[1] , C.M. IKEDA, C.Q. JUDGE, United States Naval Academy — High-speed planing boats are subjected to repeat impacts due to slamming, which can cause structural damage and injury to passengers. A first step in understanding and predicting the physics of a craft re-entering the water after becoming partially airborne is an experimental vertical drop test of a prismastic wedge (deadrise angle, $\beta = 20°$; beam, $B = 300$ mm; and length, $L = 600$ mm). The acrylic wedge was mounted to a rig allowing it to free-fall into a deep-water tank (5.2m x 5.2m x 4.2m deep) from heights $0 \leq H \leq 635$ mm, measured from the keel to the free surface. The wedge was instrumented to record vertical position, acceleration, and pressure on the bottom surface. A pressure mapping system, capable of measuring several points over the area of the thin (0.1 mm) film sensor at sampling rates up to 20 kHz, is used and compared to surface-mounted pressure transducers (sampled at 10 kHz). A high speed camera (1000 fps, resolution of 1920 x 1200 pixels) is mounted above the wedge model to record the wetted surface as the wedge descended below the free surface. The pressure measurements taken with both conventional surface pressure transducers and the pressure mapping system agree within 10% of the peak pressure values (0.7 bar, typical).

[1]Supported by the Office of Naval Research.

8:52AM G28.00005 Experimental investigation of the inception of a spilling breaker , DAN LIBERZON, Faculty of Civil and Environmental Engineering, Technion, Haifa, Israel, LEV SHEMER, School of Mechanical Engineering, Tel Aviv University, Tel Aviv, Israel — Conditions for the inception of a spilling breaker were studied in 18 m long tank. Peregrine breather-type wave train was excited to generate breaker at a desired location. Parameters of the breaker were obtained using wave gages and two synchronized 2 Mega pixel cameras operating at 60 fps. The instantaneous surface elevation in the vicinity of the breaker's crest was measured by 5 wave gages, while the local wave shape and the inception of breaking were identified from 18 Mpixel video records of the contact line shape variation at the side wall of the tank. An additional identical camera looking at the wave field from above was used to measure the velocity field in the vicinity of the breaking location using Particle Tracking Velocimetry (PTV). Floating particles with diameter of about 3 mm were used for that purpose. Both cameras were synchronized. The instantaneous crest location and velocity were determined from surface elevation fluctuations records. Actual local instantaneous crest velocities differ from both the phase and group velocities of the dominant wave and are compared with the instantaneous horizontal water velocities at various stages of waves breaking.

9:05AM G28.00006 Generation of surface waves by an underwater moving bottom: experiments and application to tsunami modeling[1] , LEONARDO GORDILLO, TIMOTHÉE JAMIN, Matière et Systémes Complexes (MSC), Univ. Paris Diderot, GERARDO RUIZ-CHAVARRíA, Facultad de Ciencias, Universidad Nacional Autónoma de México, MICHAEL BERHANU, ERIC FALCON, Matière et Systémes Complexes (MSC), Univ. Paris Diderot — Most of the ocean waves that we observe in nature are generated by processes that take place near the ocean surface. This occurs mainly because fluid layers reduce significantly the transfer of motion between the source and the free surface as the depth increases. In any case, when the disturbances at a deep source are wide and fast enough, a wave can still be generated. The archetype of this kind of process is tsunami generation: during earthquakes, the seabed of the ocean experiences a sudden net vertical displacement that can yield waves capable of flooding entire coastlines. In this talk, we will focus on laboratory experiments concerning the generation of free surface waves in a three-dimensional uniform layer whose bottom uplifts suddenly. Based on simultaneous measurements of the free surface deformation and the velocity field, we analyze the wave generation dependence on the bottom kinematics. Our results display excellent agreement with a classical linear theory of gravity waves. In addition, we develop a new theoretical approach that can be applied to improve real-time numerical simulations used by the tsunami hazard mitigation programs.

[1]Supported by the AXA Research Fund

9:18AM G28.00007 Measurements of turbulence in the airflow above surface waves[1] , FABRICE VERON, MARC BUCKLEY, University of Delaware — We present experimental results on the details of the airflow above surface gravity waves for a several wind speeds, wave ages and slopes. The bulk of the results presented were obtained from a series of laboratory experiments that took place at the University of Delaware's Air-sea interaction facility. Airflow properties within and above the viscous sublayer were obtained using PIV, and wave profiles and spectra were measured by laser-induced fluorescence. We observe direct evidence of intermittent separation of the viscous sublayer past the crest of the wind waves. The separation leads to dramatic along-wave variability in the surface viscous tangential stress which in turn may affect wave growth and the air-water momentum balance. Despite the intermittent aspect of this phenomenon, ensemble averages of the wave phase-locked velocity products suggests the airflow separation yield significant flux of vorticity away from the surface thereby generating intense mixing and momentum transport within the airflow. These results hold for wind speeds that would normally be considered low to moderate. Implications for models of air-sea momentum flux will be discussed.

[1]Funded by OCE-0850663 and OCE-0748767 from the National Science Foundation.

9:31AM G28.00008 Impact of Sea Spray on Air-Sea Fluxes[1] , FABRICE VERON, JAMES MUELLER, University of Delaware — The contributions of sea spray drops to the total air-sea exchanges of momentum, heat, and mass remain an open question. A number of factors obscure any simple quantification of their contribution: the number of drops formed at the ocean surface and the per-drop contribution to the fluxes. To estimate these per-droplet fluxes, we present results from a large number of drop trajectories, which are simulated with a recently developed Lagrangian Stochastic model adapted for the heavy drop transport and evaporation within the marine boundary layer. Then, using commonly accepted spray generation functions we present estimates of spray fluxes which account for the mediating feedback effects from the droplets on the atmosphere. The results suggest that common simplifications in previous sea spray models, such as the residence time in the marine boundary layer, may not be appropriate. We further show that the spray fluxes may be especially sensitive to the size distribution of the drops. The total effective air-sea fluxes lead to drag and enthalpy coefficients that increase modestly with wind speed. The rate of increase for the drag coefficient is greatest at moderate wind speeds, while the rate of increase for the enthalpy coefficient is greatest at higher wind speeds.

[1]Funded by grants OCE-0850663 and OCE-0748767 from the National Science Foundation.

9:44AM G28.00009 Eulerian and Lagrangian effects of surface wave on turbulence underneath , XIN GUO, LIAN SHEN, Mechanical Engineering & St. Anthony Falls Laboratory, University of Minnesota, Twin Cities — Direct numerical simulation is performed to study the effects of surface wave on underlying turbulence. In the simulations, fully nonlinear kinematic and dynamic boundary conditions are applied at the free surface. The evolution of surface elevation is obtained by advancing the kinematic boundary condition with a Runge-Kutta scheme. In the vertical direction, grid is clustered towards the free surface to ensure the boundary layers of the free surface and surface wave are fully resolved. For spatial discretization, pseudo-spectral method is used in the horizontal directions, and second-order finite difference method is used in the vertical direction. The interaction of surface wave with underlying turbulence is carefully studied in both Eulerian and Lagrangian frames. In the Eulerian frame, turbulence statistics become wave-phase dependent due to the distortion of both the free surface and the periodic wave strain field. Budget of the Reynolds normal stresses is analyzed. In the Lagrangian frame, net effect of surface wave on turbulence is identified. It is found that the net wave effect is contributed by both the Stokes drift and the correlation between the wave field and the distorted turbulence field.

9:57AM G28.00010 Breaking of waves in deep water[1] , GERARDO RUIZ-CHAVARRIA, Facultad de Ciencias, Universidad Nacional Autonoma de Mexico — The breaking of waves is a nonlinear phenomenon during which a fraction of the energy is dissipated. In the previous stage the wave undergoes a growth of its amplitude and the wave pattern is modified in the sense that the crests become more pronounced than the troughs. The breaking has been extensively studied in the case of waves approaching the shore. However, the wave breaking in deep water remains an open problem in fluid dynamics. In this work we study the wave breaking due to focusing of an initially parabolic wave front. To this end the evolution of wave is numerically investigated using a meshless code (Smoothed Particle Hydrodynamics). We present some results about the evolution of waves excited by a parabolic wave maker, among others, the growth induced by the focusing, the behavior around the Huygens' cusp and the process of wave breaking. Then, we compare the numerical results with the criteria given in the literature about the onset of breaking and we discuss how the energy dissipates, for example by the rise of short waves. In addition we compare the numerical results with data obtained in two different experiments made by our team.

[1]Author acknowledges DGAPA-UNAM by support under project IN116312, "Vorticidad y ondas no lineales en fluidos."

Monday, November 25, 2013 8:00AM - 10:10AM –

Session G30 Instability: Rayleigh-Taylor I 408 - Ranga Narayanan, University of Florida

8:00AM G30.00001 Experiments on Effects of Initial Conditions and Material Strength on Rayleigh-Taylor Instability[1] , PAMELA ROACH, Missouri S&T, ARINDAM BANERJEE, Lehigh University — The effects of initial conditions on Rayleigh-Taylor (RT) instability in an accelerated elastic-plastic solid were studied. A novel rotating wheel RT experiment that uses centrifugal forces to accelerate a two-material interface was utilized to study the effect of amplitude and wavelength on RT instability with an elastic-plastic solid. The experiment consists of a container filled with air and mayonnaise, a non-Newtonian emulsion, with an initial perturbation between the two materials. Single mode perturbations of various amplitudes and wavelengths were analyzed and results indicated the acceleration required for instability increased for both decreasing initial amplitude and wavelength. Three-dimensional interfaces were found to be more stable than two-dimensional interfaces. Critical amplitude and growth rates were compared with prior experimental results and analytical growth models. Elastic and plastic peak amplitude responses were observed for stable interfaces using a variable acceleration profile where the test section was first accelerated to slightly below the critical acceleration and then decelerated at the same rate. This exercise allowed for verification of the elastic-plastic (EP) transition process before instability was reached.

[1]Authors acknowledge financial support from DOE-LANL subcontract # 173667-1 to Lehigh University and a NSF-Graduate Research Fellowship to Pamela Roach.

8:13AM G30.00002 Mixed-mode instability of a miscible interface due to coupling between Rayleigh-Taylor and double-diffusive convective modes , JORGE CARBALLIDO-LANDEIRA, PHILIP TREVELYAN, CHRISTOPHE ALMARCHA, ANNE DE WIT, Non Linear Physical Chemistry Unit, Université Libre de Bruxelles — In a gravitational field, a horizontal interface between two miscible fluids can be buoyantly unstable because of double diffusive effects or because of a Rayleigh-Taylor instability arising when a denser fluid lies on top of a less dense one. We show here both experimentally and theoretically that, besides such classical buoyancy-driven instabilities, a new mixed mode dynamics exists when these two instabilities act cooperatively. This is the case when the upper denser solution contains a solute A, which diffuses sufficiently faster than a solute B initially in the lower layer to yield non-monotonic density profiles after contact of the two solutions. We derive analytically the conditions for existence of this mixed mode in the (R, δ) parameter plane, where R is the buoyancy ratio between the two solutions and δ is the ratio of diffusion coefficient of the solutes. We find an excellent agreement of these theoretical predic- tions with experiments performed in Hele-Shaw cells and with numerical simulations.

8:26AM G30.00003 Buoyancy Driven Mixing By Microwave Volumetric Energy Deposition , ADAM J. WACHTOR, VERONIKA MOCKO, FARZANEH F. JEBRAIL, MALCOLM J. ANDREWS, ROBERT A. GORE, Los Alamos National Laboratory — An investigation of buoyancy driven mixing of two miscible fluids due to volumetric energy deposition by microwaves is presented. The experimental setup is initially Rayleigh-Taylor stable and consists of a light, non-polar fluid at rest atop a heavier, polar fluid. Microwaves preferentially heat the polar fluid, and its density decreases due to thermal expansion. As microwave heating continues, the density of the lower fluid eventually becomes less than that of the upper fluid, thus, the system passes through the neutral stability point and becomes Rayleigh-Taylor unstable, causing buoyancy driven mixing. The evolution of the experimental design from proof-of-concept, to a customized facility designed for enhanced data collection is discussed. In addition, the fluid selection criteria found necessary for experimental success is presented. Single fluid heating experiments were performed to facilitate model development used to predict the neutral stability point and onset of buoyancy driven mixing. Results from the two-fluid mixing experiments demonstrate the capability of this novel Rayleigh-Taylor driven experiment. Particular interest is paid to the onset of buoyancy driven mixing, and atypical aspects of the experiment in the context of typical Rayleigh-Taylor driven mixing.

8:39AM G30.00004 Miscible and immiscible 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 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 induction motors and accelerated downward causing the initially stable interface to be unstable and allowing the Rayleigh-Taylor instability to develop. Forced and unforced experiments are conducted using both immiscible and miscible liquid combinations. Forced initial perturbations are 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 growth. Comparisons between miscible and immiscible mixing layer distributions are made from the resulting interface concentration profiles.

8:52AM G30.00005 An analysis of the Rayleigh-Taylor instability of thin viscous layers , E.M. DE LA CALLEJA, S. ZETINA, R. ZENIT, Universidad Nacional Autonoma de Mexico — Recently, Zetina and Zenit (2013) showed that certain textures in the early paintings of D.A. Siqueiros resulted from a hydrodynamic instability. Siqueiros invented the so-called "accidental painting" technique, which consisted in pouring layers of different color son top of each other. For the correct color combination, the dual layer became Rayleigh-Taylor unstable and mixed; the density of a paints depends on its color. In this investigation, we conducted experiments to fully understand the instability of thin viscous layers. We varied the densities, viscosities and thicknesses of the layers. We measured the size of the visible blobs and characterized their change in size with the parameters of the flow. We contrasted our observations with the predictions of a linear instability analysis of the flow. We discuss the implications of these results with modern painting techniques.

9:05AM G30.00006 Rayleigh-Taylor instability under a curved substrate , HYOUNGSOO KIM, NAIMA HAMMOUD, HOWARD STONE, Princeton University — The instability of a thin film under a curved substrate is studied experimentally. A thin film layer is uniformly coated inside a concave surface. We investigate the evolution of the liquid layer by varying the film thickness and the radius of curvature. Two typical perturbation patterns are observed; a flow perturbation in the angular direction and a periodic wavy pattern in the axial direction. These modes are observed at different Bond numbers. Although the classic Rayleigh-Taylor instability of a thin film under a flat substrate is unconditionally unstable, our experimental study highlights that the thin film is conditionally stable due to the curvature. If the aspect ratio between the thin film thickness and the radius of the curved substrate is small enough, the upside-down thin film is always stable. We compare our results with theory (P. Trinh and colleagues).

9:18AM G30.00007 Experimental measurements of velocity and density statistics in Rayleigh-Taylor instability at High Atwood numbers , DEVESH RANJAN, BHANESH AKULA, TOM FINN, Texas A&M University — Velocity statistics are measured in a Rayleigh-Taylor mixing layer at Atwood number 0.6 using the multilayer gas tunnel facility at Texas A&M University, which is capable of achieving mixing Reynolds numbers around 30000. Particle Image Velocimetry (PIV) and hot wire anemometry are used to measure the instantaneous velocities inside the mixing layer. The techniques are validated for small Atwood number and plane mixing layer experiments. The velocity statistics obtained including u_{rms},v_{rms}, $\overline{u'v'}$, $\overline{\rho'v'}$ and $\overline{\rho'^2}$ are presented and their variation across the mixing layer is also discussed. The probability density functions of the velocities, densities and their spectra are also presented.

9:31AM G30.00008 Experimental investigation of combined Rayleigh-Taylor instability and Kelvin-Helmholtz instability at different Atwood numbers , BHANESH AKULA, TOM FINN, Texas A&M University, MALCOLM ANDREWS, Los Alamos National Laboratory, DEVESH RANJAN, Texas A&M University — Combined Rayleigh-Taylor instability (RTI) and Kelvin-Helmholtz instability (KHI) is studied at three different Atwood numbers using the multilayer gas tunnel facility Texas A&M University. S3WCA (Simultaneous 3 wire and cold wire anemometry) and Particle Image Velocimetry (PIV) are used to measure instantaneous velocities and densities at different locations along and across the mix layer. High resolution digital imaging is performed during the experiments 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. Different parameters obtained from measurements including, molecular mixing parameter θ, u_{rms}, v_{rms} velocity profiles, velocity correlations, vertical turbulent mass flux $\overline{\rho' v'}$ and their effect on mixing is discussed.

9:44AM G30.00009 Efficient mixing in stratified flows: Rayleigh-Taylor instability within a stable stratification, experiments and computation , MEGAN DAVIES WYKES, University of Cambridge, ANDREW LAWRIE, University of Bristol, STUART DALZIEL, University of Cambridge — When a Rayleigh-Taylor unstable interface is confined within an otherwise stable stratification the resulting mixing efficiency can be higher than 0.75. This process has been investigated through the use of laboratory experiments, computational numerics and a simple theoretical model. Boussinesq laboratory experiments will be presented, which examine three distinct initial stratifications. Computational experiments using an ILES code have also been performed. The mixing efficiency of laboratory and computational experiments agrees very well. A theoretical model, developed to predict the size of the turbulent mixing region that grows at the unstable interface, matches the results of laboratory experiments.

9:57AM G30.00010 Experiments on the expansion wave driven Rayleigh-Taylor instability , ROBERT MORGAN, OLEG LIKHACHEV, JEFFREY JACOBS, The University of Arizona — Experiments are presented in which a diffuse interface between two gases is accelerated to generate the Rayleigh-Taylor instability. The initially flat interface is generated by the opposing flow of two test gases at matched volumetric flow rates exiting through small holes in the test section. This interface is then accelerated by an expansion wave generated by the rupturing of a diaphragm separating the heavy gas from a tank evacuated to $\sim$ 0.1atm. The expansion wave generates a very high, O(1e3g0), but non-constant acceleration acting on the interface causing the Rayleigh-Taylor instability to develop. Planar Mie scattering is employed using a planar laser sheet generated at the top of the apparatus, which illuminates smoke seeded in in a small amount of air in the heavy gas. Scattered light is then imaged using a CMOS camera operating at 12 kHz. Shadowgraphy is also used to visualize the instability using 200 mm diameter f/6.0 parabolic mirrors along with three CMOS cameras operating at 10 kHz with exposure times of 1e-6s. Perturbations are introduced by either horizontally or vertically oscillating the fluid interface to generate single-mode or random-mode perturbations respectively. Instability amplitude and growth rates are extracted and will also be presented.

Monday, November 25, 2013 8:00AM - 9:57AM –

Session G31 Porous Media Flows V: CO2 Sequestration 402 - Alfredo Soldati, University of Udine

8:00AM G31.00001 Experimental investigation of the infiltration of liquid CO2 into water-saturated, two-dimensional porous micro-models using micro-PIV , F. KAZEMIFAR, G. BLOIS, D.C. KYRITSIS, K.T. CHRISTENSEN, Univ. of Illinois — A novel experimental apparatus has been developed to study the interaction of two immiscible fluids in a two-dimensional porous micro-model. The two fluids considered are liquid CO_2 and water. This flow process is relevant to many engineering applications such as sequestration of CO_2 in geological formations as well as enhanced oil recovery operations. Saline aquifers have very high potential for geological sequestration of CO_2 based on several factors, including high capacity, economics and minimum environmental impact. In such a process, CO_2 must displace the resident fluid of the porous structure; i.e. brine. The lower viscosity and density of CO_2 compared to brine results in complex mechanisms of brine displacement. While early studies focused on qualitative observations of fluid–fluid interactions, in this study, the microscopic particle image velocimetry technique is employed to simultaneously quantify the flow fields within each fluid phase. The interface dynamics, migration and trapping mechanisms are of particular interest. In such flows, viscosity and interfacial tension are the controlling parameters which, in the vicinity of the critical point, become very sensitive to changes in pressure and temperature.

8:13AM G31.00002 Capillary pinning of immiscible gravity currents in porous media , BENZHONG ZHAO, Massachusetts Institute of Technology, CHRISTOPHER MACMINN, Yale University, HERBERT HUPPERT, University of Cambridge, RUBEN JUANES, Massachusetts Institute of Technology — Gravity currents in porous media have attracted interest recently in the context of geological carbon dioxide (CO_2) storage. Capillarity can be important in the spreading and migration of the buoyant CO_2 after injection because the typical pore size is very small, but the impact of capillarity on these flows is not well understood. Here, we study the impact of capillarity on the buoyant spreading of a finite gravity current of non-wetting fluid into a dense, wetting fluid in a vertically confined, horizontal aquifer. We show via simple, table-top experiments using glass bead packs that capillary pressure hysteresis pins a portion of the fluid-fluid interface. The horizontal extent of the pinned portion of the interface grows over time and this is responsible for ultimately stopping the spreading of the buoyant current after a finite distance. In addition, capillarity blunts the leading edge of the buoyant current. We demonstrate through micromodel experiments that the characteristic height of the nose of the current is controlled by the pore throat size distribution and the balance between capillarity and gravity. We develop a theoretical model that captures the evolution of immiscible gravity currents and predicts the maximum migration distance.

8:26AM G31.00003 Onset of nonlinear convection in transient diffusive boundary layers: application to CO_2 sequestration , NILS TILTON, AMIR RIAZ, University of Maryland, College Park — The linear stability of transient diffusive boundary layers in porous media has been studied extensively for applications to carbon dioxide sequestration. The onset of nonlinear convection, however, remains understudied because the transient base-state invalidates traditional weakly nonlinear stability methods. We study the onset of nonlinear convection using complementary asymptotic expansions and high-order direct numerical simulations (DNS). We first demonstrate by DNS that when a boundary layer is perturbed with a single Fourier mode, nonlinear mechanisms generate a zero-wavenumber response that becomes equal-order with the fundamental mode after the onset of nonlinear convection. This invalidates traditional weakly nonlinear methods that assume the zero-wavenumber response is small. Nevertheless, we demonstrate that the initial onset time of nonlinear convection can be accurately determined from a regular asymptotic expansion that is two orders-of-magnitude faster than DNS. Using the expansion, we find there is an optimal perturbation wavenumber and initial perturbation time that minimize the onset time of nonlinear convection. We obtain analytical relationships for these optimal parameters in terms of typical aquifer properties and initial perturbation magnitude.

8:39AM G31.00004 Dissolution patterns from geochemical reactions during Rayleigh-Benard convection in porous media , XIAOJING FU, LUIS CUETO-FELGUEROSO, Massachusetts Institute of Technology, DIOGO BOLSTER, University of Notre Dame, RUBEN JUANES, Massachusetts Institute of Technology — Convective mixing is an essential trapping mechanism during CO_2 sequestration in deep saline aquifers. Upon injection, buoyant CO_2 enters the geologic formation and mixes with the underlying brine, which leads to a local density increase that triggers density-driven flow; meanwhile, the presence of CO_2 disturbs the geochemical equilibrium of brine with respect to the formation, which can lead to dissolution or precipitation of carbonate minerals. Dissolution/precipitation reactions result in changes in porosity, which in turn induce changes in permeability that impact the flow dynamics. Motivated by the process of CO_2 convective mixing in deep saline aquifers, here we study the formation of rock-dissolution patterns that arise from geochemical reactions during Rayleigh-Bénard convection in porous media. We perform high-resolution simulations to analyze the interplay between the density-driven hydrodynamic instability and the formation of high-porosity channels, explain the emergence of a characteristic length scale in the dissolution channels, and quantify the impact of the channelization process on the macroscopic convection rate.

8:52AM G31.00005 Transient diffusive boundary layers in heterogeneous porous media , DON DANIEL, AMIR RIAZ, University of Maryland, College Park — The onset of convection of gravitationally unstable transient, diffusive boundary layers in saline aquifers has been extensively investigated due to its importance in carbon dioxide sequestration. However, due to theoretical complexities, the onset of convection in heterogeneous porous media with span-wise variation in permeability (varying crosswise to direction of gravity) has been relatively less explored. Unlike homogeneous porous media, we demonstrate that the onset of convection in such heterogeneous media is simultaneously triggered by a combination of several individual span-wise perturbation modes. Using a combination of linear stability analysis and numerical simulations, we obtain the dominant two dimensional global perturbation structures for different spatially varying permeability fields. We compare our results with previous literature for homogeneous porous media, and conclude whether the onset of convection in heterogeneous media is enhanced or delayed.

9:05AM G31.00006 Stability of High Rayleigh-Number Equilibrium Solutions of the Darcy–Oberbeck–Boussinesq Equations[1] , BAOLE WEN, University of New Hampshire, LINDSEY CORSON, University of Strathclyde, GREGORY CHINI, University of New Hampshire — There has been significant renewed interest in dissolution-driven convection in porous layers owing to the potential impact of this process on carbon dioxide storage in terrestrial aquifers. In this talk, we present some numerically-exact equilibrium solutions to the porous medium convection problem in small laterally-periodic domains at high Rayleigh number Ra. The "uni-cellular" equilibrium solutions first found by Corson and Chini (2011) by solving the steady Darcy–Oberbeck–Boussinesq equations are recovered and, in the interior (i.e. away from upper and lower boundary layers), are shown to have the same horizontal-mean structure as the "heat-exchanger" solutions identified by Hewitt et al. (2012). Secondary stability analysis of the steady solutions is performed, and implications for high-Ra porous medium convection are discussed.

[1]Funding from NSF Award 0928098 is gratefully acknowledged.

9:18AM G31.00007 Dissolution driven convection for carbon dioxide sequestration: the stability problem , SHREYAS MANDRE, XINJUN GUO, Brown University, ANJA SLIM, Schlumberger-Doll Research Center — The dissolution-driven convection in porous media is potentially a rate limiting process for sequestering carbon dioxide in underground aquifers. Super critical carbon dioxide introduced in the aquifer is lighter than the water that fills the surrounding porous rock, and hence quickly rises to the top. However, the solution of carbon dioxide in water is heavier than water. Hence, as the layer of carbon dioxide dissolves in the water, convection may ensue. The threshold criteria for convection is obscured by the continually changing background density profile as the carbon dioxide diffuses through the pores. Commonly used techniques such as frozen coefficient analysis or non-modal theories using transient amplifications yield substantially different results for the threshold, which has been the cause of a debate in the scientific community. We present a general theory for the linear stability of non-autonomous systems and apply it to dissolution driven convection. The theory unifies the classical modal stability theory using eigenvalues, the non-modal approaches using optimal growth of energy and the frozen coefficient analysis. We settle the debate, and demonstrate the existence of a threshold time for convection to commence.

9:31AM G31.00008 Hazards from a massive release of CO2 such as the 1986 Lake Nyos event , DIANA SHER, ANDREW WOODS, BP Institute University of Cambridge — We report a series of new experiments exploring the dynamics of gravity currents in which a volume of dense fluid is released from a lock gate and spreads along a flume into the ambient fluid. Using new experimental data, we develop a model for the evolution of the current with time, and in particular, the evolution of the flow passing through a given point with time. We apply this to interpret the hazards associated with a large release of dense gas, such as occurred in 1986 at Lake Nyos, Cameroon, when about 0.1 cubic km of CO2 was released from the lake and travelled down a valley as a dense gravity current.

9:44AM G31.00009 Impact of wettability on two-phase displacement patterns in granular media , RUBEN JUANES, MATHIAS TROJER, MICHAEL SZULCZEWSKI, Massachusetts Institute of Technology, RAN HOLTZMAN, Hebrew University — Two phase flow in porous media controls many natural processes like geological CO2 sequestration, enhanced oil recovery, water infiltration in soil, and methane venting from organic-rich sediments. While the wetting properties of rocks can vary drastically, the effect of wettability on fluid displacement in porous media remains poorly understood. Here, we study experimentally how wettability affects the fluid-fluid displacement pattern in rigid granular media within the capillary and viscous fingering regime. We inject a less viscous fluid into a thin bed of glass beads, initially fully saturated with a more viscous one. By keeping all control parameters constant and changing the contact angle of the substrate systematically, we visually explore and quantify the impact of the wetting properties on the invasion morphology. For fixed capillary number, we show that the invasion pattern becomes more stable as the contact angle increases (i.e., as we transition from drainage to imbibition) both in the capillary-fingering and in the viscous-fingering regime. We quantify the dependence of the lengthscale of the instability on contact angle, and propose a mechanistic pore-scale model that explains the macroscopic observations.

Monday, November 25, 2013 8:00AM - 9:44AM –

Session G32 Granular Flows I: Impact, Locomotion and Drag 403 - Amos Winter, Massachusetts Institute of Technology

8:00AM G32.00001 Impact response of shear thickening suspensions , ERIC BROWN, Yale University, OKTAR OZGEN, MARCELO KALLMANN, BENJAMIN ALLEN, University of California, Merced — Dense suspensions of hard particles such as cornstarch in water exhibit shear thickening, in which the energy dissipation rate under shear dramatically increases with increasing shear rate. Recent work has established that in steady-state shear this phenomena is a result of a dynamic jamming of the particles in suspension. Several dynamic phenomena observed in such suspensions have long been assumed to be a consequence of this shear thickening; strong impact resistance, the ability of a person to run on the fluid surface, fingering and hole instabilities under vibration, and oscillations in the speed of sinking of an object in the fluid. However, I will present results of experiments consisting of an indenter impacting a dense suspension which demonstrate that the strong impact resistance cannot be explained by existing models for steady-state shear thickening. I will show these dynamic phenomena can be reproduced by graphical simulations based on a minimal phenomenological model in which the fluid has a stiffness with a dependence on velocity history. These and other recent results suggest a need for new models to understand the dynamic phenomena associated with shear thickening fluids.

8:13AM G32.00002 Impact in dense granular suspensions: crucial role of dilatancy and pore pressure feedback[1] , YOEL FORTERRE, IUSTI CNRS Aix-Marseille University, J. JOHN SOUNDAR JEROME, NICOLAS VANDENBERGHE, LAURENT DUCHEMIN, IRPHE CNRS Aix-Marseille University — We investigate the impact of a rigid sphere onto a granular paste made of non-buoyant glass beads mixed with a liquid under gravity. We show that the initial volume fraction of the granular packing has a critical influence on the impact behavior. For loose packing, the ball sinks in the granular medium as in a liquid, giving rise to a collapsing cavity and a central jet as observed with fine powders in air. By contrast, for dense packing, the ball stops as soon as it hits the surface and its kinetic energy is almost instantly dissipated. We interpret this "liquid-solid" transition as the volume fraction change by a coupling between dilatancy effects and the liquid pore pressure during the impact. Dynamic pore pressure measurements and a simple diphasic model taking into account dilatancy support this mechanism. Our results show that "shear-thickening-like" phenomena in granular suspensions can arise from transient diphasic coupling rather than from the intrinsic rheology of the material.

[1]This work was supported by ANR through the program No. ANR-11-JS09-005-01.

8:26AM G32.00003 Dimensional analysis scaling of impact craters in unconsolidated granular materials , DAVID R. DOWLING, Mechanical Engineering, University of Michigan, Ann Arbor MI 48109-2133, THOMAS R. DOWLING, Skyline High School — Dimensional analysis is a general technique for determining how the independent parameters that describe physical phenomena must be arranged to produce dimensionally self-consistent results. This presentation describes how dimensional analysis may be successfully applied to the formation of impact craters produced by dropping spherical objects into a bed of unconsolidated granular material. The experiment is simple and safe, and laboratory results for different impact energies (0.001 to 1.6 J), seven different spheres (masses from 4 to 64 grams, diameters from 1.0 to 4.3 cm), and two different dry granular materials (granulated sugar, and playground sand) may be collapsed to a single power-law using parametric scaling determined from dimensional analysis. Thus, impact crater formation may provide a useful validation test for simulations of granular material dynamics. Interestingly, the scaling law shows that the impacting sphere's diameter is not a parameter. And, the resulting power law can be extrapolated, with some success, over more than 16 orders of magnitude to produce an independent estimate of the impact energy that formed the 1.2-km-diameter Barringer Meteor Crater in northern Arizona.

8:39AM G32.00004 Drag reduction due to interstitial air in a granular medium , TESS HOMAN, DEVARAJ VAN DER MEER, Physics of Fluids, University of Twente — The force experienced by an object while it penetrates a pre-fluidized sand bed strongly depends on the ambient air pressure. In this work we experimentally investigate the influence of interstitial air by systematically varying the penetration velocity and the ambient air pressure and measuring the resulting force required to push the intruder into the sand bed. Counterintuitively, we find that for the intruder to move faster through the bed a *lower* force is required. We hypothesize that, while the object moves down, sand in front of the intruder is compacted and the air in this compactified region is trapped. At higher penetration velocities air has no time to move out of the way causing a pressure build-up in front of the ball which leads to drag reduction. To test this hypothesis, we perform experiments at reduced ambient air pressures and find that indeed the dependence on the intruder velocity disappears: The measured force is constant and equal to the value of the drag found in the quasi-static limit, which emphasizes the role of air.

8:52AM G32.00005 The Mechanics of Localized Fluidization Burrowing , AMOS WINTER, MIT Dept. of Mechanical Engineering — This presentation will focus on the granular mechanics and critical timescales related to localized fluidization burrowing, a digging method inspired by the Atlantic razor clam (*Ensis directus*). The animal uses motions of its valves to locally fail and then fluidize surrounding soil to reduce burrowing energy and drag. The characteristic contraction time to achieve fluidization can be determined from substrate properties. The geometry of the fluidized zone is dictated by the coefficient of lateral earth pressure and friction angle of the soil. Calculations using full ranges for these parameters indicate that the fluidized zone is a local effect, occurring between 1–5 body radii away from the animal. The energy associated with motion through fluidized substrate – characterized by a depth-independent density and viscosity – scales linearly with depth. In contrast, moving through static soil requires energy that scales with depth squared. For engineers, localized fluidization offers a mechanically simple and purely kinematic method to dramatically reduce energy costs associated with digging. This concept is demonstrated with RoboClam, an *E. directus*-inspired robot. Using a genetic algorithm to find optimal digging kinematics, RoboClam has achieved localized fluidization burrowing performance comparable to that of the animal, with a linear energy-depth relationship, in both idealized granular glass beads and *E. directus*' native cohesive mudflat habitat.

9:05AM G32.00006 Reversibility in locomotion in granular media[1] , WILLIAM SAVOIE, DANIEL GOLDMAN, Georgia Tech — A recent study of a self-deforming robot [Hatton et al, PRL, 2013] demonstrated that slow movement in dry granular media resembles locomotion in low Re fluids, in part because inertia is dominated by friction. The study indicated that granular swimming was kinematically reversible, a surprise because yielding in granular flow is irreversible. To investigate if reciprocal motions lead to net displacements in granular swimmers, in laboratory experiments, we study the locomotion of a robotic "scallop" consisting of a square body with two flipper-like limbs controlled to flap forward and backward symmetrically (a flap cycle). The body is constrained by linear bearings to allow motion in only one dimension. We vary the the flapping frequency f, the body/flipper burial depth d, and the number of flaps N in a deep bed of 6 mm diameter plastic spheres. Over a range of f and d, the $N = 1$ cycle produces net translation of the body; however for large N, a cycle produces no net translation. We conclude that symmetric strokes in granular swimming are irreversible at the onset of self-deformation, but become asymptotically reversible.

[1]work supported by NSF and ARL

9:18AM G32.00007 Sidewinding as a control template for climbing on sand , HAMIDREZA MARVI, Georgia Institute of Technology, CHAOHUI GONG, Carnegie Mellon University, NICK GRAVISH, Georgia Institute of Technology, JOSEPH MENDESLON, Georgia Institute of Technology and Zoo Atlanta, ROSS HATTON, Oregon State University, HOWIE CHOSET, Carnegie Mellon University, DANIEL GOLDMAN, DAVID HU, Georgia Institute of Technology — Sidewinding, translation of a limbless system through lifting of body segments while others remain in static contact with the ground, is used by desert-dwelling snakes like sidewinder rattlesnakes *Crotalus cerastes* to locomote effectively on hard ground, rocky terrain, and loose sand. Biologically inspired snake robots using a sidewinding gait perform well on hard ground but suffer significant slip when ascending granular inclines. To understand the biological organisms and give robots new capabilities, we perform the first study of sidewinding on granular media. We vary the incline angle ($0 < \theta < 20°$) of a trackway composed of desert sand. Surface plate drag measurements reveal that as incline angle increases, downhill yield stresses decrease by 50%. Our biological measurements reveal that the animals double the length of the contact region as θ increases; we hypothesize that the snakes control this contact to reduce ground shear stress and avoid slipping. Implementing the anti-slip motion in a snake robot using contact patch modulation enables the robot to ascend granular inclines.

9:31AM G32.00008 A predictive, nonlocal rheology for granular flows , KEN KAMRIN, MIT, DAVID HENANN, Brown University — We propose a continuum model for flowing granular matter and demonstrate that it quantitatively predicts flow and stress fields in many different geometries. The model is constructed in a step-by-step fashion. First we compose a relation based on existing granular rheological approaches (notably the "inertial" granular flow rheology) and point out where the resulting model succeeds and where it does not. The clearest missing ingredient is shown to be the lack of an intrinsic length-scale. To tie flow features more carefully to the characteristic grain size, we compose a nonlocal model that includes a new size-dependent term (with only one new material parameter). This new nonlocal model resolves some outstanding questions in the granular flow literature — of note, it is the first model to predict all features of flows in split-bottom cell geometries, a decade-long open question in the field. In total, we will show that this new model, using three material parameters, quantitatively matches the flow and stress data from over 160 experiments in several different geometries.

Monday, November 25, 2013 8:00AM - 10:10AM –

Session G33 Drops VII: Wetting and Spreading 404 - Alban Sauret, Princeton University

8:00AM G33.00001 Shapes of non-circular drops on inclined hysteretic surfaces , NACHIKETA JANARDAN, MAHESH PANCHAGNULA, Indian Institute of Technology Madras — The shapes of non-circular drops on inclined hysteretic surfaces have been studied experimentally. The drops have an initially elliptical triple line and are formed by allowing two circular drops to coalesce. The drop is initially at rest on a horizontal substrate at different orientations to the tilt axis. This substrate is then tilted and the drop is allowed to move down the inclined substrate. The moving and sliding angles are measured as a function of the initial triple line topology and surface characteristics. The moving angle is the first critical inclination angle at which the triple line is on the verge of being deformed. The sliding angle is the second critical inclination angle at which the entire drop is in a state of impending motion. It is seen that the drop shape, sliding and moving angle are dependent on the initial conditions and the initial orientation of the drop. The moving angle, which is an indication of the hysteresis force attempting to keep the drop from sliding, is shown to scale with the triple line length, as well as the contact angle hysteresis. In addition, the configurations of the drops during the evolution process are studied to establish the mechanism for the sliding process.

8:13AM G33.00002 Liquid spreading in the partial wetting regime , AMIR A. PAHLAVAN, MICHAEL CHEN, LUIS CUETO-FELGUEROSO, GARETH H. MCKINLEY, RUBEN JUANES, Massachusetts Institute of Technology — The flow of thin films over flat surfaces has been the subject of much theoretical, experimental and computational research [D. Bonn et al., Rev. Mod. Phys., 2009]. Using the lubrication approximation, the classical mathematical model for these flows takes the form of a nonlinear fourth-order PDE, where the fourth-order term models the effect of surface tension [e.g. H. E. Huppert, Nature, 1982]. This classical model effectively assumes that the film is perfectly wetting to the substrate, whereas partial wetting is responsible for stopping the spread of a liquid puddle. Here, we present experiments of (large-volume) liquid spreading over a flat horizontal substrate in the partial wetting regime, and characterize the four spreading regimes that we observe. We develop a macroscopic phase-field model of thin-film flows that naturally accounts for the dynamic contact angle. Our model therefore permits describing thin-film flows without invoking a precursor film, leading to compactly-supported solutions that reproduce the spreading dynamics and the static equilibrium configuration observed in the experiments.

8:26AM G33.00003 Drag on Sessile Drops[1] , ANDREW J.B. MILNE, University of Alberta/MIT, BRIAN FLECK, DAVID NOBES, DEBJYOTI SEN, University of Alberta, ALIDAD AMIRFAZLI, York University, UNIVERSITY OF ALBERTA MECHANICAL ENGINEERING COLLABORATION — We present the first ever direct measurements of the coefficient of drag on sessile drops at Reynolds numbers from the creeping flow regime up to the point of incipient motion, made using a newly developed floating element differential drag sensor. Surfaces of different wettabilities (PMMA, Teflon, and a superhydrophobic surface (SHS)), wet by water, hexadecane, and various silicone oils, are used to study the effects of drop shape, and fluid properties on drag. The relation between drag coefficient and Reynolds number (scaled by drop height) varies slightly with liquid-solid system and drop volume with results suggesting the drop experiences increased drag compared to similar shaped solid bodies due to drop oscillation influencing the otherwise laminar flow. Drops adopting more spherical shapes are seen to experience the greatest force at any given airspeed. This indicates that the relative exposed areas of drops is an important consideration in terms of force, with implications for the shedding of drops in applications such as airfoil icing and fuel cell flooding. The measurement technique used in this work can be adapted to measure drag force on other deformable, lightly adhered objects such as dust, sand, snow, vesicles, foams, and biofilms.

[1]The authours acknowledge NSERC, Alberta Innovates Technology Futures, and the Killam Trusts

8:39AM G33.00004 ABSTRACT WITHDRAWN –

8:52AM G33.00005 Static and Dynamic Contact Angles of Immersed Ferrofluid Droplets , SOUVICK CHATTERJEE, Virginia Tech, DIPANWITA BHOWMIK, Jadavpur University, ACHINTYA MUKHOPADHYAY, IIT Madras, RANJAN GANGULY, University of Illinois Chicago — Ferrofluid plug driven micro-pumps are useful for manipulating micro-volume of liquids by providing remote actuation using a localized magnetic field gradient. Inside a microchannel, the ferrofluid experiences combined actions of different relevant body forces. While the pressure, viscous and magnetic forces can be estimated using established techniques, the surface tension force requires information about the contact angle between the ferrofluid and glass capillary wall. We address this phenomenon through experimental characterization of static and dynamic contact angles of oil based ferrofluid (EFH3) droplets on glass surface immersed in pure or surfacted distilled water. The equilibrium static contact angle is found to significantly reduce in presence of a magnetic field. Dynamic contact angles are measured through high-speed imaging as the ferrofluid droplets slide along an inclined glass surface. Variation of contact angle hysteresis, which falls outside the Hoffmann Tanner equation for this case, is also investigated as a function of contact line velocity. A strong dependence is found between the contact angle hysteresis and the wetting time. Findings of the work is useful for designing ferrofluid plug-driven microfluidic plugs for different lab-on-a-chip applications.

9:05AM G33.00006 Volume-filtered Surface Forces for the Simulation of Contact Lines , GERRY DELLA ROCCA, GUILLAUME BLANQUART, California Institute of Technology — Although contact lines are present in many industrial processes, there is no universal agreement how to implement these singularities in numerical simulations. Often a dynamic contact-angle law is applied with a slip boundary condition at the contact line. However, most slip boundary conditions require realistic slip lengths much smaller than the largest length scale. At a sufficient numerical grid resolution, this multi-scale problem have a prohibitive computational expense. In this study, volume-averaged source terms are constructed for both a contact-angle restoring surface force and the viscous dissipation in the vicinity of the contact line. Closure terms are proposed for the unresolved slip length scales. Unlike previous studies, no a-priori angle relation is applied and the contact angle evolves naturally from the explicitly represented physics. Simulations of fluid displacement in a channel and droplet spreading are demonstrated with this new framework.

9:18AM G33.00007 Relaxation of contact-line singularities solely by the Kelvin effect and apparent contact angles for isothermal volatile liquids in contact with air[1] , ALEXEY REDNIKOV, PIERRE COLINET, Universite Libre de Bruxelles - TIPs, CP 165/67 — The contact (triple) line of a volatile liquid on a flat solid is studied theoretically. Like with a pure-vapor atmosphere [Phys. Rev. E 87, 010401, 2013], but here for isothermal diffusion-limited evaporation/condensation in the presence of an inert gas, we rigorously show that the notorious contact-line singularities (related to motion or phase change itself) can be regularized solely on account of the Kelvin effect (curvature dependence of the saturation conditions). No disjoining pressure, precursor films or Navier slip are in fact needed to this purpose, and nor are they taken into consideration here ("minimalist" approach). The model applies to both perfect (zero Young's angle) and partial wetting, and is in particular used to study the related issue of evaporation-induced contact angles. Their modification by the contact-line motion (either advancing or receding) is assessed. The formulation is posed for a distinguished immediate vicinity of the contact line (the "microregion"), the corresponding problem decoupling to leading order, here up to one unknown coefficient, from what actually happens at the macroscale. The lubrication approximation (implying sufficiently small contact angles) is used in the liquid, coupled with the diffusion equation in the gaz phase.

[1]Supported by ESA and BELSPO PRODEX and F.R.S.-FNRS.

9:31AM G33.00008 X-ray imaging technique for studying contact-line statics and dynamics of drops on soft substrates , SU JI PARK, JI SAN LEE, JUN HO LEE, JINKYUNG KIM, POSTECH, BYUNG MOOK WEON, Sungkyunkwan University, JUNG HO JE, POSTECH — When a drop sits on a soft surface, its surface tension deforms the soft material and creates a wetting ridge. X-ray microscopy is useful to measure the shape of the ridge with high spatial and temporal resolutions. This technique allows us to directly image ridge-growth dynamics in real time. We find that the ridge-tip formation is actually asymmetric and independent of substrate stiffness and growth dynamics. From this situation, we directly measure the solid surface stresses. Our approach is a general technique that can be used to measure surface stresses for soft materials within a wide stiffness range. Finally, we suggest a general framework of the wetting behaviors on a soft solid with the combination of Young's and Neumann's laws in macroscopic and microscopic scales, respectively. X-ray microscopy would be useful for further understanding of contact-line dynamics on soft materials.

9:44AM G33.00009 Multiscale approach to studying super-spreading: molecular dynamics and continuum-level models[1] , PANAYIOTIS THEODORAKIS, ERICH MULLER, RICHARD CRASTER, OMAR MATAR, Imperial College London — We study surfactant-assisted spreading of fluid droplets on solid substrates. The "super-spreading" problem is a prime example of complex behavior exhibited by such systems, which is not fully-understood. Continuum-level models disregard molecular architecture effects or the specific interactions between the building blocks of the system; hence, they are unable to provide a physical reasoning for the super-spreading mechanism at the molecular level. Molecular dynamics (MD) simulations are restricted to small systems, and are, therefore, unable to provide a continuum description of spreading. Hence we employ a multi-scale modeling approach to study the problem. We use the Statistical Associating Fluid Theory to estimate the intermolecular potentials between the solvent and the surfactant particles using a top-down coarse-grained approach. As a result, we have achieved quantitative matching of our simulations to experimental macroscopic thermophysical properties. Based on these interactions, we perform MD simulations by taking into account, the molecular architecture of surfactants, and we estimate the relevant microscopic parameters and boundary conditions for use in our continuum description. Results from our multiscale approach will be presented at the meeting.

[1]EPSRC Grant (EP/J010502/1)

9:57AM G33.00010 Effects of the size of the domain in the evolution of thin films , JUAN MANUEL GOMBA, CIFICEN - UNCPBA - CONICET, JONATAN RAUL MAC INTYRE, CIFICEN - UNCPBA - CONICET (Argentina), CARLOS ALBERTO PERAZZO, Departamento de Física y Química, Universidad Favaloro, Solís 453, 1078, Buenos Aires, Argentina — We investigate theoretically the possible final stationary configurations that can be reached by a laterally confined uniform film of liquid. The liquid is under the action of gravity, surface tension and the molecular interaction with the solid substrate. The governing parameters of the problem are the initial thickness of the fluid, the size of the recipient that contains the liquid, and a dimensionless number which quantifies the relative strength of gravity with respect to the molecular interaction. The uniform film is always a possible final state, and depending on the value of the parameters may exist up to 3 different additional final states, each one consisting in a drop with a thin precursor film. We derive analytical expressions for the energy of these possible final configurations, and from this we analyze which one is indeed reached. We conclude that the fluid may show three different behaviors after perturbation: the system recovers its initial shape for any perturbation, the system evolves towards a drop (if more than one is possible, the final state corresponds is the one with the thinnest precursor film) for any perturbation, or the system ends as a uniform film or a drop depending on the details of the perturbation.

Monday, November 25, 2013 8:00AM - 10:10AM –

Session G34 Drops VIII: Fragmentation 405 - Nikolai Priezjev, Michigan State University

8:00AM G34.00001 Explosive fragmentation , ALEXANDRE VLEDOUTS, Aix Marseille Université, IRPHE, Marseille, France, JOSÉ GRAÑA-OTERO, School of Aeronautics, Universidad Politécnica de Madrid, Spain, JOEL QUINARD, NICOLAS VANDENBERGHE, EMMANUEL VILLERMAUX, Aix Marseille Université, IRPHE, Marseille, France — We report on an experiment consisting in forcing the fast radial expansion of a spherical liquid shell. The shell is formed by the capillary pinch off of a water thin annular jet surrounding a jet of reactive gaseous mixture at ambient pressure. The encapsulated gas in the resulting water bubble is a mixture of Hydrogen and Oxygen in controlled relative proportions, which is ignited by a laser plasma aimed at the center of the bubble. The strongly exothermic combustion of the mixture induces the expansion of the hot burnt gas, pushing the shell radially outwards in a violently accelerated motion. That motion triggers the instability of the shell, developing thickness modulations ultimately piercing it in a number of holes. The capillary retraction of the holes concentrates the liquid constitutive of the shell into a web of ligaments, whose breakup leads to stable drops. We document the overall process, from the kinematics of the shell initial expansion, to the final drops size distribution as a function of the composition of the gas mixture and bubble shell thickness.

8:13AM G34.00002 Drop Size Distributions of Aerated Liquid Jets injected in Subsonic Crossflow , ADEGBOYEGA ADEBAYO, KHALED SALLAM, Oklahoma State University, KUO-CHENG LIN, Taitech, CAMPBELL CARTER, AFRL — An Experimental investigation of the breakup of aerated liquid jets in subsonic crossflow is described. Test conditions include crossflow Mach numbers of 0.3 and 0.6, Gas-to-liquid ratio of 0%, 4%, and 8%. Double pulsed digital holography was used to investigate the spray characteristics at downstream distances of 25, 50, and 100 jet diameters. The holograms are analyzed using image-processing algorithms to yield information about the drop sizes, drop velocities, and mass fluxes. Different drop size distributions are tested and compared including Rosin-Rammler distribution, log-normal distribution, and Simmons' universal root-normal distribution.

8:26AM G34.00003 An ultrasonic scrubber: enhanced removal of particles by water sprays via ultrasonic excitation , J.R. SAYLOR, WEIYU RAN, Clemson University, R. GLYNN HOLT, Boston University — Sprays are commonly used to remove pollutant particles in smokestacks, to reduce coal dust levels in mines, and in dust abatement applications. For typical conditions, sprays work poorly on particles having a diameter on the order of a micron, which is also the particle size most deleterious to the human lung. The acoustic radiation force can be used to move particles and drops, and we hypothesized that by forcing a particle laden flow and a spray into an ultrasonic standing wave field, particles and drops would be concentrated, thereby increasing the effectiveness of particle removal by sprays. Experimental data is presented in the form of scavenging coefficients for micron scale particles that supports this hypothesis. Also discussed is whether increased scavenging by ultrasonics is due to particle/drop interactions particle/particle interactions, or both.

8:39AM G34.00004 Atomization in Sparkling Fireworks , CHIHIRO INOUE, University of Tokyo, MITSUO KOSHI, Yokohama National University, HIROSHI TERASHIMA, TAKEHIRO HIMENO, TOSHINORI WATANABE, University of Tokyo, SPARKLING FIREWORKS TEAM — The physics behind the beauty of sparkling fireworks has not been clarified yet due to a lack of coherent visualization results. In the present study, atomization process in sparkling fireworks is elucidated by using a high-speed video camera. In the first-half sequence of the fireworks, the fireball repeatedly expands, bursts, and shrinks due to the high pressure gas inside the fireball. In contrast, in the last-half sequence, the bubbly fireball slightly deforms, and small bubbles burst on the fireball. A scenario of droplets generation is as follows: a liquid thread extends from the bottom of the bursting fireball, and fragments into droplets. Thus the droplets originate from inside the fireball rather than from its surface.

8:52AM G34.00005 Free-fall of Water Drops Generated in the Laboratory for Rainfall Simulations[1] , FIRAT TESTIK, NASIM CHOWDHURY, MATHEW HORNACK, ABDUL KHAN, Glenn Department of Civil Engineering, Clemson University, Clemson, SC — The shape and fall velocity evolution of water drops that are falling freely were studied using high-speed imaging. Digital image processing techniques were utilized to measure drop characteristics. Water drops of three target diameters were generated using needles placed at 12 different stations (approximately 1 m apart vertically for a fall distance of 12 m) of a stairwell and sequential high-speed drop images were obtained at the bottom floor. Our experimental observations indicated that generated drops underwent three distinct regions during free-fall. The first region, where different oscillation modes prevail, is influenced by the source conditions. The drop oscillations are absent in the second region and the shapes of the drops transition towards equilibrium shapes. Through the third region, the drops experience free-fall at equilibrium shapes and terminal velocities and possess characteristics similar to those of raindrops in an actual rain event. The ranges of the different fall regions were delineated based upon our experimental observations. The results of this study are useful in (i) determining the fall distance required for laboratory simulation of rainfall and (ii) studying the shape - fall velocity coupling of raindrops in the laboratory.

[1]This material is partially based upon work supported by the National Science Foundation under Grant No. AGS 1144846

9:05AM G34.00006 Interfacial Instabilities on a Droplet , MAZIYAR JALAAL, Department of Mechanical Engineering, The University of British Columbia, Vancouver, BC, Canada, KIAN MEHRAVARAN, School of Engineering, The University of British Columbia, Kelowna, BC, Canada — The fragmentation of droplets is an essential stage of several natural and industrial applications such as fuel atomization and rain phenomena. In spite of its relatively long history, the mechanism of fragmentation is not clear yet. This is mainly due to small length and time scales as well as the non-linearity of the process. In the present study, two and three-dimensional numerical simulations have been performed to understand the early stages of the fragmentation of an initially spherical droplet. Simulations are performed for high Reynolds and a range of relatively high Weber numbers (shear breakup). To resolve the small-scale instabilities generated over the droplet, a second-order adaptive finite volume/volume of fluids (FV/VOF) method is employed, where the grid resolution is increased with the curvature of the gas-liquid interface as well as the vorticity magnitude. The study is focused on the onset and growth of interfacial instabilities. The role of Kelvin-Helmholtz instability (in surface wave formation) and Rayleigh-Taylor instability (in azimuthal transverse modulation) are shown and the obtained results are compared with the linear instability theories for zero and non-zero vorticity layers. Moreover, the analogy between the fragmentation of a single drop and a co-axial liquid jet is discussed. The current results can be used for the further development of the current secondary atomization models.

9:18AM G34.00007 Local stability of a fluid interface near a zero-vorticity point[1] , YU-HAU TSENG, ANDREA PROSPERETTI, Johns Hopkins University — There are many examples of fluid interfaces which give rise to small-scale structures in uniform or nearly uniform flows: "skirted" bubbles, filaments trailing rising drops and bubbles or forming at the tip of coaxial jets and others. It is argued that these phenomena are due to a peculiar instability in the neighborhood of a zero-vorticity point (or line). A local stability analysis supports this conjecture and is further illustrated by some numerical results.

[1]Supported by a grant from BP/The Gulf of Mexico Research Initiative through the University of Texas Marine Science Institute (DROPPS consortium: "Dispersion Research on Oil: Physics and Plankton Studies")

9:31AM G34.00008 Ligament breakup without surface tension , LIONEL VINCENT, LAURENT DUCHEMIN, STÉPHANE LE DIZÈS, EMMANUEL VILLERMAUX, Aix Marseille Université, IRPHE, Marseille, France — We study the breakup of an axisymmetric low viscosity liquid volume (ethanol and water), held by surface tension on supporting rods, when subject to a violent axial stretching. One of the rods is promptly set into a fast motion, either with constant acceleration, or constant velocity. In both cases, a thin ligament is withdrawn from the initial liquid volume, which eventually breaks-up at time t_b, leaving a liquid mass m attached to the moving rod. We find that the breakup time and entrained mass are related by $t_b \sim \sqrt{m/\sigma}$, where σ is the liquid surface tension. For a constant acceleration γ, and although the overall process is driven by surface tension, t_b is surprisingly found to be independent of σ, while m is inversely proportional to γ. The case with constant velocity will be considered too.

9:44AM G34.00009 Influence of Geometry on Instability: Breakup of fluid strips with square-wave perturbations[1] , KYLE MAHADY, SHAHRIAR AFKHAMI, LOU KONDIC, New Jersey Institute of Technology — Recent work[2] has demonstrated experimentally and computationally that an originally flat structure with an imposed nonlinear square-wave perturbation applied to the edges could break up in a variety of ways. In this talk we will report on the results of a computational study that centers on solving the Navier-Stokes equations using a volume of fluid approach. We focus on exploring the details of the breakup mechanism and on the influence of the initial fluid shape on the instability development. One unexpected result is the finding that the initial geometry may strongly influence the outcome, and in particular lead to closely spaced array of drops. The size and spacing of the drops is found to be strongly influenced by nonlinear stages of the evolution, and cannot be predicted based on the Rayleigh-Plateau instability mechanism.

[1]Supported in part by the NSF grant CBET-1235710.
[2]Roberts, N., Fowlkes, J., Mahady, K., Afkhami, S., Kondic, L. and Rack, P. *ACS Applied Materials and Interfaces* **2013**, 5, 4450.

9:57AM G34.00010 The viscous Savart sheet , EMMANUEL VILLERMAUX, Aix Marseille Université, IRPHE, Marseille, France, VIOLAINE PISTRE, Techni-Process, 13790, France, HENRI LHUISSIER, Aix Marseille Université, IRPHE, Marseille, France — We study the viscous version of the planar Savart sheet problem, using an impacting liquid jet up to 300 times more viscous than water. Two surprising observations are made, contrasting with the traditional case introduced by Savart where viscosity plays no role: First, if the radius of a viscous sheet is typically reduced compared to the water case for a given jet radius and impacting velocity, the smooth/flapping transition is delayed, allowing for smooth sheet radii substantially bigger than those permitted with water at large impacting Weber number. Second, the drop size distribution is bimodal, with a substantial fraction of the drops having a very small, well defined diameter. We understand these two facts in terms of an additional model experiment, and simple physical arguments.

Monday, November 25, 2013 8:00AM - 10:10AM –

Session G35 Chaos, Fractals, and Dynamical Systems I: Coherent Structures 406 - Shawn Shadden, University of California, Berkeley

8:00AM G35.00001 Lagrangian Coherent Structures and their application to ocean transport , THOMAS PEACOCK, MICHAEL ALLSHOUSE, MIT — The approach of Lagrangian Coherent Structures (LCSs) holds great promise for improved understanding of flow transport in the ocean, with potentially significant application to scenarios such as improved decision making strategies for pollution events. We present some improved methodology for identifying, refining and classifying LCSs in data sets and assess the utility of the approach through a number of case studies.

8:13AM G35.00002 Coherent structures in reacting flows[1] , JOHN MAHONEY, KEVIN MITCHELL, University of California, Merced — Our goal is to characterize the nature of reacting flows by identifying important "coherent" structures. We follow the recent work by Haller, Beron-Vera, and Farazmand which formalized the the notion of lagrangian coherent structures (LCSs) in fluid flows. In this theory, LCSs were derived from the Cauchy-Green strain tensor. We adapt this perspective to analogously define coherent structures in *reacting* flows. By this we mean a fluid flow with a reaction front propagating through it such that the propagation does not affect the underlying flow. A reaction front might be chemical (Belousov-Zhabotinsky, flame front, etc.) or some other type of front (electromagnetic, acoustic, etc.). While the recently developed theory of burning invariant manifolds (BIMs) describes barriers to front propagation in time-periodic flows, this current work provides an important complement by extending to the aperiodic setting.

[1]The present work was supported by the US National Science Foundation under grants PHY- 0748828 and CMMI-1201236

8:26AM G35.00003 An extension of shear and strain LCS concepts to higher dimensions[1] , SIAVASH AMELI, SHAWN C. SHADDEN, University of California, Berkeley — A framework is presented for the extension of strain and shear barrier concepts to R^n. The concept of shear barrier was introduced by Haller & Beron-Vera [Physica D 241 2012] in R^2. The framework presented herein also generalizes to normally hyperbolic, or strain LCS, as introduced by Haller [Physica D 240 2011]. We use a projection operator approach to define Lagrangian shear strain and Lagrangian normal strain vector fields from the Cauchy-Green strain tensor. These Lagrangian strain vector fields are the basis for defining maximal shear LCS, and maximal and minimal strain LCS. Criteria for shear and strain LCS are natural analogs, helping to unify these concepts.

[1]This work was supported by the National Science Foundation, award number 1047963.

8:39AM G35.00004 Lagrangian Descriptors: A Method for Revealing Phase Space Structures of General Time Dependent Dynamical Systems[1] , ANA M. MANCHO, ICMAT, CSIC, STEPHEN WIGGINS, University of Bristol, JEZABEL CURBELO, ICMAT, UAM, CAROLINA MENDOZA, Universidad Politecnica de Madrid — Lagrangian descriptors are a recent technique which reveals geometrical structures in phase space and which are valid for aperiodically time dependent dynamical systems. We discuss a general methodology for constructing them and we discuss a "heuristic argument" that explains why this method is successful. We support this argument by explicit calculations on a benchmark problem. Several other benchmark examples are considered that allow us to assess the performance of Lagrangian descriptors with both finite time Lyapunov exponents (FTLEs) and finite time averages of certain components of the vector field ("time averages"). In all cases Lagrangian descriptors are shown to be both more accurate and computationally efficient than these methods.

[1]We thank CESGA for computing facilities. This research was supported by MINECO grants: MTM2011-26696, I-Math C3-0104, ICMAT Severo Ochoa project SEV-2011-0087, and CSIC grant OCEANTECH. SW acknowledges the support of the ONR (Grant No. N00014-01-1-0769).

8:52AM G35.00005 Experimental Three Dimensional Lagrangian Coherent Structures of Inertial Particles in Flows , SAMUEL RABEN, SHANE ROSS, PAVLOS VLACHOS, Virginia Tech — Finite Time Lyapunov Exponents (FTLE) are a powerful and increasingly popular tool for describing mixing and transport in both turbulent and laminar flow fields. FTLEs provide a measure of the exponential rate of divergence or convergence of Lagrangian particle trajectories and can be used both experimentally and numerically to describe a flow field, which may have a high degree of spatiotemporal complexity. While primarily used to describe single-phase flow behavior some works have attempted to account for inertial particles by modeling the particles' motion through simulations. This procedure can provide insight, but does not provide direct information about the true observable inertial particle trajectories. This work provides a method to more directly determine FTLEs from experimental data for inertial particles through the use of particle tracking velocimetry (PTV) without any a-priori assumptions about particle motion. We show, in a turbulent 3D flow field, how FTLE for various particles sizes can be computed with out numerical integration and how separating the particles effects the resulting FLTE field. This work can provide future insight into multiphase flow research and the study of inertial particle motion.

9:05AM G35.00006 Inertial particle dynamics: Coherent structures in the presence of the Basset–Boussinesq memory term , MOHAMMAD FARAZMAND, GEORGE HALLER, ETH Zurich — We present an equivalent formulation of the Maxey–Riley equation in the presence of the Basset–Boussinesq memory term. A physical advantage of this formulation is that it reveals drag- and pressure-type forces within the memory term. The computational advantage of the new form is that it turns the Maxey–Riley equation from an implicit differential equation into an explicit one, enabling the use of classic numerical schemes in its solution. We further simplify the Maxey–Riley equation for small particles by deriving its reduction to its attractor. The reduced equation obtained in this fashion reveals that the memory term is asymptotically of the order of $\mathrm{St}^{3/2}$, with St being the Stokes number. This explains recent numerical findings on the relative importance of the Basset–Boussinesq term. Finally, we compute inertial Lagrangian coherent structures (ILCS) for vortex shedding behind a cylinder. The reduced ILCS closely capture the full inertial dynamics while providing significant savings in computational cost and complexity.

9:18AM G35.00007 Experimental and Numerical Study of Transition to Turbulence in a Kolmogorov-Like Flow[1] , BALACHANDRA SURI, JEFFREY TITHOF, RADFORD MITCHELL JR., ROMAN GRIGORIEV, MICHAEL SCHATZ, Center for Nonlinear Science and School of Physics, Georgia Institute of Technology — Recent theoretical advances suggest that turbulence can be characterized using exact unstable solutions of the Navier Stokes equations, called Exact Coherent Structures (ECS). Due to their experimental accessibility and theoretical tractability, two-dimensional flows provide an ideal setting for the exploration of turbulence from a dynamical systems perspective. In our talk, we present a combined numerical and experimental study of electromagnetically driven flows in a shallow layer of electrolyte. Our experimental results include the sequence of bifurcations the flow undergoes en route to becoming weakly turbulent. We discuss the effects of boundaries on the flow structure. On the numerical front, we present results from a 2D DNS, comparing them with the experiment. Also, in the weakly turbulent simulation of the flow, we search for exact coherent structures and present a few we have identified.

[1]This work is supported in part by the National Science Foundation under grants No. CBET-0853691, CBET-0900018, and CMMI-1234436

9:31AM G35.00008 Characterizing the dynamics of unsteady planar flows through the topology of coherent-structure-based trajectories , MARK STREMLER, PRADEEP RAO, SHANE ROSS, Virginia Tech — There has been significant development in the identification of coherent structures associated with Lagrangian transport. Methods include Lagrangian Coherent Structures (LCS), which identifies transport barriers with minimal flux between regions, and Almost Invariant Sets (AIS) or the related Finite-Time Coherent Sets (FTCS), which identify the coherent regions that are separated by transport barriers. These methods are valuable tools for identifying key features of complex spatio-temporal transport at given instants in time. Understanding how the time-dependent interaction of these structures relates to the global characteristics of transport in the system has proven a more difficult task. We present evidence that space-time trajectories embedded in coherent structures, which we identify via AIS or FTCS, can be used to describe the global structure of transport in the flow. For sufficiently complex flows, these trajectories 'braid' about one another, and the topology of this braid can be directly correlated with chaos in the system. We investigate the connection between the occurrence of braiding AIS/FTCS trajectories and the exponential stretching of material lines associated with chaos in several example flows, including lid-driven cavity flow and the double gyre flow.

9:44AM G35.00009 ABSTRACT WITHDRAWN –

9:57AM G35.00010 Thermal coherent sets and heat transfer in chaotic laminar flows , SHIBABRAT NAIK, Mitsubishi Electric Research Labs and Virginia Tech, PIYUSH GROVER, Mitsubishi Electric Research Labs — The relation between the chaotic nature of the advection flow field and heat transfer in laminar flow heat exchangers is known to be subtle. We use the Perron-Frobenius transfer operator approach to analyze thermal transport in a coiled tube with 3D laminar flow and Dirichlet thermal boundary condition. The usual advection-only transfer operator is combined with a finite-difference diffusion operator via an operator-splitting technique. We compute various coherent sets of this approximate advection-diffusion operator. These coherent sets correspond to the important "thermal structures" which govern the heat transfer in this problem. This analysis gives an insight into the effect of chaotic advection field on the heat transfer performance of such devices. We study the dependence of heat transfer enhancement factor on Peclet number.This transfer operator based analysis could lead to systematic geometric optimization of micrometer sized heat exchangers.

Monday, November 25, 2013 8:00AM - 10:10AM –

Session G36 Microfluids: Drops/Bubbles 407 - Taehun Lee, City College of New York

8:00AM G36.00001 A co-flow-focusing monodisperse microbubble generator , JIAMING ZHANG, ERQIANG LI, SIGURDUR SIGURDUR, King Abdullah University of Science and Technology — Here we report the design and fabrication of a simple and inexpensive microfluidic device based on micro-scope glass slides and two tapered glass capillaries, for generating monodisperse microbubbles. The first capillary that used for transporting gas, was heated and pulled to have a sharp tapered tip around 2 microns, and was inserted into the second capillary, with its sharp tip aligned to the converging-diverging throat of the second capillary. This configuration provides a smooth, small gas flow rate, as well as a high velocity gradient at the gas outlet. By varying liquid flow rates and viscosities, highly monodisperse microbubbles with diameter range from 3.5 to 50 microns have been successfully produced, at a rate up to 50 kHz. A simple scaling law based on capillary number and flow rate ratios, used to predict bubble size, is also proposed.

8:13AM G36.00002 Critical behavior of droplet breakup in T-junction microchannels , VOLKERT VAN STEIJN, DUONG HOANG, LUIS PORTELA, CHRIS KLEIJN, MICHIEL KREUTZER, Delft University of Technology — The critical behavior of droplet breakup in T-junction mirochannels is studied using three-dimensional numerical simulations. Two scenarios can happen when a droplet flows into a T-junction: (i) if the flow is strong enough, it breaks into two daughter droplets and (ii) otherwise, it drifts away into one branch of the T-junction owing to flow perturbations. Whether a droplet breaks or not is determined by the ratio between two timescales: breakup time and drifting time. Symmetric-boundary-condition simulations allow us to study the breakup time without any flow perturbations, thus to accurately compute the critical capillary number below which the droplet does not break. We study the drifting using full-T-junction simulations, identifying three phases in drifting process: (i) an exponential drifting, (ii) a transition phase and (iii) a linear drifting. Combining the understanding of the breakup and drifting behavior, we found that the critical capillary number below which the droplet drifts away increases more than 10% with respect to the one obtained in free-perturbation flow systems.

8:26AM G36.00003 ABSTRACT WITHDRAWN –

8:39AM G36.00004 Tipstreaming from the rear of surfactant laden droplets traveling through a microchannel , TODD MOYLE, LYNN WALKER, SHELLEY ANNA, Carnegie Mellon University Department of Chemical Engineering — Microscale tipstreaming is a hydrodynamic phenomenon able to generate submicron sized droplets in a microfluidic device. The tipstreaming process results in the generation of a thin thread from a highly curved interface. In this work, we present observations of tipstreaming occurring at the rear of droplets translating along a microchannel. Drops are formed in a flow focusing geometry at geometry-controlled formation conditions. The drops then accelerate in the exit channel due to the addition of continuous phase liquid from two intersecting channels. Upon acceleration, the droplets form a highly curved tip at the rear and begin to shed a stream of tiny drops. The distance between the acceleration point and the location downstream at which tipstreaming occurs depends on surfactant concentration, drop size, and flow rate of the added continuous phase liquid. We examine the effect of these parameters on the tipstreaming process. Because tipstreaming occurs downstream of the acceleration point, after the drop has attained a new steady state velocity, we hypothesize that the adsorption of additional surfactant on the interface is the primary factor driving the onset of tipstreaming. We use these measurements to probe the timescale for surfactant adsorption to the droplet interface.

8:52AM G36.00005 Gas bubble formation and its pressure signature in T-junction of a microreactor[1] , SHAHRAM POUYA, MANOOCHEHR KOOCHESFAHANI, Michigan State University — The segmented gas-liquid flow is of particular interest in microreactors used for high throughput material synthesis with enhanced mixing and more efficient reaction. A typical geometry to introduce gas plugs into the reactor is a T-junction where the dispersed liquid is squeezed and pinched by the continuous fluid in the main branch of the junction. We present experimental data of time resolved pressure along with synchronous imaging of the drop formation at the junction to show the transient behavior of the process. The stability of the slug regime and the regularity of the slug/plug pattern are investigated in this study.

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

9:05AM G36.00006 Early microfluidic dissolution regime of CO_2 bubbles in viscous oils[1] , MARTIN SAUZADE, THOMAS CUBAUD, Stony Brook University — We investigate the initial dynamical behavior of dissolving micro-bubbles composed of carbon dioxide gas in highly viscous silicone oils over a range of flow rates and pressure conditions. Microfluidic periodic trains of monodisperse CO_2 bubbles are used to probe the interrelation between bubble dissolution and high-viscosity multiphase flows in microgeometries. The effective mass diffusion flux across the interface is measured by tracking individual bubbles and monitoring their shape as they experience a size reduction. The initial steady mass flux is characterized using a dissolution coefficient that depends on the fluids physicochemical properties. Our findings show the possibility to control and exploit the interplay between capillary and mass transfer phenomena with highly viscous fluids in small-scale systems.

[1]This work is supported by NSF (CBET- 1150389)

9:18AM G36.00007 The Physical Mechanisms Governing Drop Coalescence: Models vs Experiments , JAMES SPRITTLES, University of Oxford, YULII SHIKHMURZAEV, University of Birmingham — The dominant physical mechanisms in the coalescence of liquid drops are identified by utilizing recent advances in experimental and computational techniques that resolve unprecedentedly small spatio-temporal scales. To do so, the predictions of both the "conventional" model and the (singularity-free) interface formation model, where the dynamics of an "internal interface" trapped between the two bulk phases takes a finite time to disappear, before the conventional model takes over, are compared to experimental measurements on microfluidic scales of the very initial moments of coalescence. Using the full numerical solution of the problem in the framework of each of the two models, we show that the recently reported electrical measurements are better described by the interface formation model. As a by product of our results, the range of validity of scaling laws proposed for the phenomenon is established, with inconsistencies in previous works rectified and particular attention paid to quantifying the external gas' effect on the dynamics. Finally, a new scaling law developed for the inertial regime is shown to capture experimental data from the literature remarkably well.

9:31AM G36.00008 Optofluidic droplet coalescence on a microfluidic chip[1] , JIN HO JUNG, KYUNG HEON LEE, KANG SOO LEE, HYUNJUN CHO, BYUNG HANG HA, GHULAM DESTGEER, HYUNG JIN SUNG, KAIST — Coalescence is the procedure that two or more droplets fuse during contact to form a larger droplet. Optofluidic droplet coalescence on a microfluidic chip was demonstrated with theoretical and experimental approaches. Droplets were produced in a T-junction geometry and their velocities and sizes were adjusted by flow rate. In order to bring them in a direct contact of coalescence, optical gradient force was used to trap the droplets. A theoretical modeling of the coalescence was derived by combining the optical force and drag force on the droplet. The analytical expression of the optical force on a sphere droplet was employed to estimate the trapping efficiency in the ray optics regime. The drag force acting on the droplet was calculated in terms of the fluid velocity, viscosity and the geometrical parameters of a microfluidic channel. The droplet coalescence was conducted in a microfluidic setup equipped with a 1064 CW laser, focusing optics, a syringe pump, a custom-made stage and a sCMOS camera. The droplets were successfully coalesced using the optical gradient force. The experimental data of coalescence were in good agreement with the prediction.

[1]This work was supported by the Creative Research Initiatives program (No.2013-003364) of the National Research Foundation of Korea (MSIP).

9:44AM G36.00009 Constrained Energy Minimization of a Pinned Droplet on an Inclined Plate , MICHIEL MUSTERD, VOLKERT VAN STEIJN, CHRIS R. KLEIJN, MICHIEL T. KREUTZER, Department of Chemical Engineering, Faculty of Applied Sciences, Delft University of Technology, Julianalaan 136, 2628 BL, Delft, the Netherlands — A long standing problem is the prediction of the maximum volume of a droplet that can hang on an inclined plate without rolling off. A key issue in this prediction is to understand the deformation of the droplet. We show that the common assumptions of a fixed droplet base or a shape at global energy minimum result in significant errors. We study droplets on a inline using locally constrained energy minimization. The initial shape of the droplet and maximum and minimum attainable contact angles hereby put constraints on the energy minimization. This results in a history dependence of the droplet behavior before roll-off, but surprisingly, a universal behaviour of the front-to-back baselength of the droplet at roll-off. This universal behavior can be predicted from equilibrium droplet shapes on a horizontal surface and understood from energy landscapes for a 2D droplet.

9:57AM G36.00010 The way to reduce electrical charge of a droplet dispensed from a pipette tip[1] , DONGWHI CHOI, HORIM LEE, DO JIN IM, DONG SUNG KIM, POSTECH — Recently, we reported that a conventional pipetting always makes a charged droplet by spontaneous electrical charging process. The charge amount depends on the constituents of the droplet, on coating material of pipette tip and on atmospheric humidity. We clarified that this natural electrification of a droplet is originated from the charge separation between a droplet and pipette tip surface. The electrical interaction between charged droplet hanging on the end of the pipette tip and the pipette tip inner surface makes the droplet hard to detach from the pipette tip. To suggest the way to suppress the electrification phenomenon, we investigate the influence of the polymer composition on the amount of the charge of the droplet. The Faraday cup method is performed to measure the charge amount of the droplet. The result can be used to reduce charge amount of a droplet dispensed from the micropipette tip effectively.

[1]This work was supported by the Mid-career Researcher Program No. 2011-0029454 funded by the Korea government (MEST).

Monday, November 25, 2013 10:30AM - 12:27PM –

Session H1 Jets III: Round, Liquid and Impinging 323 - James J. Feng, University of British Columbia

10:30AM H1.00001 Bifurcation and Turbulent transitions of round jets , PHILIPPE BARDET, AMY MCCLENEY, The George Washington University — An experimental round jet is created by controlling the axial momentum injection rate, with the resulting water jet discharging freely into a large tank. The evolution and the transition to turbulence of round jets into a quiescent fluid are examined for Reynolds number ranging from 1,000 to 10,000. The boundary layer roll-up, merging of vortices, and turbulence transition of the jets natural instabilities are tracked from the nozzle exit. Near field measurements of the flow structures are observed using PLIF through azimuthal dye injection and PIV. These types of jets are observed both in nature and industrial applications. Understanding the early transition steps of a jet in detail is important in the view of efficient turbulence control. Modification of the natural fluid flow can lead to the delay in transition when turbulence is harmful or to promote instability when better mixing is beneficial. This study focuses on improving the understanding of jet flow structures for the development of flow manipulation.

10:43AM H1.00002 The effect of viscosity gradients on the stability of the turbulent round jet , RYAN KEEDY, JAMES RILEY, ALBERTO ALISEDA, University of Washington — The effect of viscosity differences between the ambient and injected fluids on a high Reynolds number round jet is poorly understood and has been largely ignored in stability analyses of this canonical shear flow. When viscosity gradients are present at the mixing interface between the two fluids, the jet behavior can be significantly affected. A new set of linear stability equations, which account for differences in jet and ambient viscosities, have been developed to study the growth of spatial disturbances. The equations are shown to reproduce results found in the literature for constant viscosity. Eigenvalue analysis is carried out to evaluate the predicted growth rates and unstable wavelengths as a function of the dominant variables: frequency, momentum thickness, etc. Experimental results obtained in a high Reynolds number (10^5) round jet facility, with a submerged jet issuing vertically in a large, essentially unconfined water tank were compared with the parametric study of the linear stability analysis results. Jet viscosity in our experiments was modified by altering the viscosity of the miscible injected fluid over a range of the viscosity ratio, $\mu_{jet}/\mu_{ambient} > 1$ (by adding glycerol) and $\mu_{jet}/\mu_{ambient} < 1$ (by raising the temperature).

10:56AM H1.00003 Time-resolved imaging studies of adjacent liquid jet formation[1] , JULIA YANG, Carnegie Mellon University, FREDERIK BRASZ, CRAIG ARNOLD, Princeton University, ARNOLD GROUP COLLABORATION — Laser-induced jetting of liquids is an area of interest in fluid dynamics due to its versatile range of applications in printing and patterning. In this work, we use time-resolved imaging to examine the formation of two adjacent liquid jets produced by separate laser pulses of similar energy. A laser pulse is absorbed within a polymer layer coated with ink, forming a rapidly expanding blister that induces a liquid jet. The time delay and spatial separation between pulses are varied, and for close enough proximities, the second jet exhibits changes in propagation direction. As the separation between pulses decreases, the jets intertwine and form one long, twisted jet or a single large jet. The time-resolved images are also compared with simulation results, which reveal similar trends. This scenario of two adjacent liquid jets provides insight into high repetition rate printing and the limitations of separation and time delay between pulses.

[1]This work is funded by a MRSEC grant (DMR-0819860) and a NSF REU site grant (DMR-1156422).

11:09AM H1.00004 Global stability of gravitationally stretched capillary jets[1] , MARIANO RUBIO-RUBIO, ALEJANDRO SEVILLA, Departamento de Ingeniería Térmica y de Fluidos, Universidad Carlos III de Madrid, Spain, JOSÉ MANUEL GORDILLO, Departamento de Ingeniería Aeroespacial y Mecánica de Fluidos, Universidad de Sevilla, Spain — We analyze the global linear stability of capillary jets stretched by gravity both experimentally and theoretically, extending the work by Sauter & Buggisch (J. Fluid Mech. vol. 533, 2005, pp. 237-257). Our results reveal the essential stabilizing role played by the axial curvature of the jet, the latter effect being especially relevant for injectors with a large diameter. The theoretical description, based on the one-dimensional mass and momentum equations retaining the exact expression for the interfacial curvature, accurately predicts the onset of jet self-excited oscillations experimentally observed for wide ranges of liquid viscosity and injector diameter. The marginal self-sustained oscillations observed in the experiments are shown to correspond to the excitation of the leading global mode of the jet. The model developed in the present work shows better agreement with the experimental jetting-dripping transition events than those available in the literature, thus allowing us to conclude that, surprisingly, the size of the steady threads produced at a given distance from the exit can be reduced by increasing the nozzle diameter. The proposed formulation allows to describe the inviscid limit, and experiments are being performed to study this distinguished case.

[1]Supported by Spanish MINECO under projects DPI 2011-28356-C03-01 and DPI 2011-28356-C03-02.

11:22AM H1.00005 Global frequency response analysis of gravitationally stretched liquid jets[1] , PAULA CONSOLI-LIZZI, WILFRIED COENEN, ALEJANDRO SEVILLA, Área de Mecánica de Fluidos, Dpto. de Ingeniería Térmica y de Fluidos, Universidad Carlos III de Madrid, Spain — The convective capillary break-up of freely falling axisymmetric jets of Newtonian liquid is theoretically studied with a one-dimensional description of the mass and momentum conservation equations. Instead of using the classical quasi-parallel assumption in the stability analysis, here we compute the global linear response of the flow to harmonic inputs at the exit of the jet, allowing us to predict its break-up length in cases where the base flow is not slender. Our theory compares favourably with recent experiments by Javadi *et al.* (PRL 110, 144501, 2013), who measured the break-up length of unforced liquid jets of several viscosities. From the physical point of view, our main finding is that the meniscus region near the injector outlet, where the jet experiences the strongest axial stretching, delays the growth of capillary disturbances due to a *spatial* counterpart of the kinematic stabilizing mechanism firstly described by Tomotika (Proc. Roy. Soc. 153, 1936) in a *temporal* setting.

[1]Supported by Spanish MINECO under project DPI 2011-28356-C03-02.

11:35AM H1.00006 Auto-ejection of liquid jets and drops from capillary tubes , HADI MEHRABIAN, JAMES J. FENG[1], Department of Chemical and Biological Engineering, University of British Columbia — Capillary imbibition through a tube and nozzle assembly can built enough momentum to eject droplets. Such an auto-ejection process is studied using Cahn-Hilliard diffuse-interface simulations to capture the dynamic contact angle, interface deformation and drop pinch-off. The breakup process is studied and a criterion for ejection is proposed. We investigate the dependence of this criterion on geometric parameters and fluid properties. Finally, we estimate the size of the produced droplets and compare the numerical results with experiments.

[1]Department of Mathematics, University of British Columbia

11:48AM H1.00007 Surface Pressure Fluctuations Produced by an Axisymmetric Impinging Jet: Generation Mechanisms[1] , MALEK AL-AWENI, AHMED NAGUIB, Michigan State University — This study is motivated by understanding the mechanisms leading to unsteady surface pressure generation in impinging jet flows. Employing an extensive database of concurrent time-resolved flow visualization and time series from a surface-embedded microphone array, two dominant mechanisms are found to affect the space-time evolution of the pressure within the wall-jet zone: vortex-wall and vortex-vortex interaction. To gain deeper insight into these mechanisms, two closely-related model problems are studied computationally using Fluent. The problems involve the impingement of a single or two axisymmetric vortex rings on a flat wall. The resulting spatio-temporally resolved computations are used in conjunction with Possion's equation for pressure to investigate the nature of the pressure-generating sources, their relative importance, and their relation to the observed surface pressure signature. The findings provide significant information towards realization of efficient, structure-based models for computing the unsteady wall pressure in impinging jets.

[1]Partly funded by NSF grant OISE-0611984 and Libyan-North American Scholarship program

12:01PM H1.00008 Surface Pressure Fluctuations Produced by an Axisymmetric Impinging Jet: Spatio-Temporal Characteristics[1] , AHMED NAGUIB, MALEK AL-AWENI, Michigan State University — This is the second of a sequence of two presentations concerned with understanding the nature and generation mechanisms of the unsteady surface pressure in impinging jet flows. In the first presentation, the mechanisms influencing the evolution of the surface pressure are studied by examining instantaneous realizations obtained from time-resolved flow visualization and concurrent surface-embedded microphone array measurements; along with numerical simulations of related model problems. In this presentation, the focus is on examining the statistical importance and persistence of these mechanisms by comparing knowledge obtained from the instantaneous analysis to that resulting from inspection of conditional spatio-temporal surface-pressure behaviors, frequency-wavenumber spectra and other statistical measures. Results are presented for surface-pressure measurements at a Reynolds number based on jet diameter of approximately 7000. Dependence of the results on the spacing between the impingement wall and the jet as well as the jet impingement angle is also considered.

[1]Partly funded by NSF grant OISE-0611984 and Libyan-North American Scholarship program

12:14PM H1.00009 Effects of density, velocity gradient, and compressibility on side-jet formation in round jets with variable density , AKINORI MURAMATSU, Department of Aerospace Engineering, College of Sience and Technology, Nihon University — When a low density gas compared with the ambient gas is discharged from a round nozzle, side jets that are radial ejections of jet fluid are generated at the initial region of the jet. The density ratio between the jet fluid and the ambient fluid is a main parameter for the side-jet formation. Since the side-jet formation is also related to the instability of shear layer, it depends on the velocity gradient of the shear layer in the jet. The velocity gradient is evaluated by a ratio of the momentum thickness and the nozzle diameter at the nozzle exit. Compressibility suppresses the instability and the generation of the side jets. The compressibility is evaluated by a Mach number, which is a ratio defined by an issuing velocity of the jet and a sound velocity in the ambient fluid. Influence of these three parameters on the side-jet formation was examined experimentally. The density ratio and momentum thickness ratio were varied from 0.14 to 1.53, and from 14 to 155, respectively. The Mach number was varied to 0.7. Existence of side jets was confirmed by flow visualization using a laser sheet. Domains for the side-jet formation by the density ratio, the momentum thickness ratio, and the Mach number were determined.

Monday, November 25, 2013 10:30AM - 12:27PM –

Session H2 Particle-Laden Flows V: DNS and Non-Spherical Particles 324 - Ismail Celik, West Virginia University

10:30AM H2.00001 The effect of particle rotation in multi-particle flow simulations[1] , ADAM SIERAKOWSKI, ANDREA PROSPERETTI, Johns Hopkins University — In multi-particle flow simulations, particle rotation is difficult to calculate and is often imprecisely accounted for or ignored altogether. We examine the effect of these procedures on the overall flow characteristics through large systems of particles when the particle center is fixed and either allowed or not to rotate. We use a newly developed GPU-centric implementation of the Physalis method for the solution of the Navier-Stokes equations in the presence of finite-size spheres. We investigate periodic systems of more than 100 randomly-distributed particles at Reynolds numbers up to 100. By considering flow characteristics such as mean velocity and pressure drop, we shed light on the importance of including particle rotation effects in large particle-flow simulations.

[1]Study supported by NSF grant CBET 1258398.

10:43AM H2.00002 Numerical Investigation of Cloud Droplet Growth via Collision Coalescence: One Step Approach[1] , HOSSEIN PARISHANI, University of Delaware, ORLANDO AYALA, Old Dominion University, BOGDAN ROSA, Institute of Meteorology and Water Management, Poland, LIAN-PING WANG, University of Delaware — Growth of inertial particles and droplets in a turbulent flow is a critical step in a wide range of applications. It is known that cloud turbulence could make a substantial impact on the growth of cloud droplets by collision and coalescence. Reade and Collins (2000) performed a DNS study of one step growth of coagulating particles in turbulence. They found that the limiting solutions of zero or infinite St numbers are not capable of describing the dynamics of finite-inertia particles. In this talk we extend their work to include the effects of gravity and flow Reynolds number on growth of droplets in turbulence. Starting from an initially monodisperse distribution of particles, we study how turbulent collision coalescence affects the particle size distribution. The simulations are performed with a 256^3 grid resolution and $O(10^6)$ droplets of radii ranging from 10 to 60 microns. We obtain particle size distributions for a range of flow Reynolds numbers to study the effect of flow Reynolds number on the size distribution of inertial particles. The one-step results are compared to those from the kinetic collection equations using gravitational and turbulent collision kernels.

[1]This work is partially supported by NSF.

10:56AM H2.00003 Effect of ambient flow inhomogeneity on drag forces on a sphere at finite Reynolds numbers , JUNGWOO KIM, Seoul National University of Science and Technology, S. BALACHANDAR, University of Florida, HYUNGOO LEE, Korea Atomic Energy Research Institute — For studies on particle-laden flows involving particle transport and dispersion, the prediction capability of hydrodynamic forces on the particle in a non-uniform flow is one of the central issues. However, existing analytical expressions and empirical correlations are mainly made based on the homogeneous flow conditions such as uniform or uniform shear flows. Therefore, the objective of this study is to investigate the effect of flow inhomogeneity on drag forces on a sphere at finite Reynolds numbers. To do so, we perform direct numerical simulations of flow over a sphere in an inhomogeneous flow. In this study, we consider three different kinds of the inhomogeneous flows: cosine, hyperbolic cosine and hyperbolic secant profiles. The Reynolds number of the sphere based on the freestream velocity and sphere diameter is 100. The present simulations show that the quasi-steady drag forces in inhomogeneous flows are reasonably estimated by standard drag law based on the relative velocity if the fluid velocity seen by the particle is evaluated by surface average. The results support Loth and Dorgan (2009)'s proposed formula. In the final presentation, the effect of ambient flow inhomogeneity on drag forces would be presented in more detail.

11:09AM H2.00004 Particle dispersion in stably stratified open channel flow , SALVATORE LOVECCHIO, Dipartimento di Ingegneria Elettrica, Gestionale e Meccanica, University of Udine, FRANCESCO ZONTA, ALFREDO SOLDATI, Centro Interdipartimentale di Fluidodinamica e Idraulica and Dipartimento di Energetica e Macchine, University of Udine — Many geophysical flows are influenced by stable stratification effects. In terrestrial water bodies, the vertical distribution of temperature produces a thermocline (a region where large gradients occur) which strongly influences mixing. In this study we analyse the effect that the formation of the thermocline has on particle dispersion in stably stratified turbulence in an open channel flow using Direct Numerical Simulation and Lagrangian Particle Tracking. The parameter that characterizes the physical problem is given by the ratio Gr/Re_τ^2, where Gr is the Grashof number and Re_τ the Reynolds number. This parameter represents the relative importance of buoyancy and inertia (namely of stratification). We perform a parametric study, considering different stratification levels (i.e, different values of Gr/Re_τ^2) and particles with different inertia. Preferential concentration is quantified using the correlation dimension and Voronoi diagrams. Results indicate that the thermocline in the upper flow layers influences the dynamics of the coherent flow structures by reducing the frequency with which upwelling/downwelling motions of fluid are formed. This in turn decreases particle dispersion and segregation at the flow surface.

11:22AM H2.00005 Inertial Range Scaling in Rotations of Long Rods in Turbulence[1] , GREG VOTH, SHIMA PARSA[2], Wesleyan University — We measure the rotational statistics of neutrally buoyant rods with lengths $2.8 < l/\eta < 72.9$ in turbulence. For particles with length in the inertial range, we derive a scaling relationship for the mean square rotation rate, $\langle \dot{p}_i \dot{p}_i \rangle \propto l^{-4/3}$ and show that measurements approach this scaling. Deviations from the proposed scaling are explained as the effect of dissipation range scales. The correlation time of the Lagrangian autocorrelation of rod rotation rate scales as the turn over time of eddies of the size of the rod. Measuring rotational dynamics of single long rods provides a new way to access the dynamics of turbulence at fixed spatial scale in a frame advected with the flow.

[1]Supported by NSF grant DMR-1208990,
[2]Currently at SEAS, Harvard University.

11:35AM H2.00006 Alignment of vorticity and rods with Lagrangian fluid stretching in turbulence[1] , RUI NI, GREG VOTH, Wesleyan University — Stretching in continuum mechanics is naturally described using the Cauchy-Green strain tensors. These tensors quantify the stretching experienced in a Lagrangian reference frame, which provides a powerful way to study interesting processes that involve stretching, such as vortex stretching and alignment of anisotropic particles. We integrate the velocity gradient tensor from direct numerical simulation of isotropic turbulence to obtain the Cauchy-Green strain tensor. We find that the preferential alignment between anisotropic particles and vorticity is because both of them tend to align with the strongest stretching direction, defined by the maximum eigenvector of the left Cauchy-Green strain tensor. In particular, anisotropic particles approach almost perfect alignment with the strongest stretching direction. The alignment of vorticity with the stretching direction is weaker, but still much stronger than previously observed alignment of vorticity with the eigenvectors of the velocity gradient tensor.

[1]Work supported by the NSF grant DMR-1208990.

11:48AM H2.00007 On the compaction of fibers in a flow and the formation of sea balls , PATRICE LE GAL, GAUTIER VERHILLE, IRPHE - CNRS - Aix Marseille University — Sea balls found on Mediterranean beaches are made of Posidonia fibers which aggregate due to the sea motions. To understand the mechanism of aggregation and compaction of these structures, we have studied the distribution of sizes and masses of these balls. We show that the pdfs are very close to log-normal distributions which suppress some formation mechanisms such as fibers aggregation one by one (that would give Poisson distributions) or random clustering or fragmentation (that would lead to Gamma distributions). Then, we present an experimental investigation on the dynamic of aggregation of fibers by waves generated in a tank. This experiment underlines the importance of the fiber rigidity.

12:01PM H2.00008 Rotation of rigid fibers in wall shear turbulence[1] , CRISTIAN MARCHIOLI, ALFREDO SOLDATI, University of Udine — In this paper we examine the rotation of rigid fibers with different elongation and inertia in turbulent channel flow, focusing on the effect of local shear and turbulence anisotropy. Statistics of the fiber angular velocity, Ω, are extracted from DNS of turbulence at shear Reynolds number $Re_\tau = 150$ coupled with Lagrangian tracking of prolate ellipsoidal fibers with Stokes number $1 < St < 100$, and aspect ratio $1 < \lambda < 50$. Results for mean and fluctuating angular velocities show that elongation is important for fibers with small inertia ($St \leq 5$ in the present study). At larger inertia, elongation has an impact on rotation only in the streamwise and wall-normal directions. In the channel center, the Lagrangian autocorrelation coefficients of Ω and corresponding rotational turbulent diffusivities match the exponential behavior predicted by the theory of homogeneous dispersion. Also, the PDF of fiber angular velocities is generally close to Gaussian, indicating that fiber rotation away from solid walls can be modeled as a Ornstein-Uhlenbeck diffusion process at stationary state. In the strong shear region, fiber anisotropy adds to flow anisotropy to induce strong deviations on fiber rotational dynamics with respect to spherical particles.

[1]Support from COST Action FP1005 “Fiber suspension flow modelling” is gratefully acknowledged.

12:14PM H2.00009 Surface singularities of nanorod laden droplets in magnetic field[1] , KONSTANTIN KORNEV, ALEXANDER TOKAREV, Clemson University, WAH-KEAT LEE, Brookhaven National Lab — Magnetic nanorods are attractive materials enabling assembly, ordering, control, and reconfiguration of different magnetic lattices within milliseconds in milliTesla magnetic fields. In this talk we will show a new physical principle of self-assembly of magnetic nanorods into singular cusps at the droplet surface. These singularities can be formed on demand not deforming the entire droplets by taking advantage of the magneto-static interactions between nanorods in non-uniform magnetic field. Using X-ray phase contrast imaging and scaling analysis we will explain the behavior of magnetic nematics and their interactions with the droplet surface.

[1]The authors are grateful for the financial support of the National Science Foundation through Grant EFRI 0937985, and of the Air Force Office of Scientific Research through Grant FA9550-12-1-0459.

Monday, November 25, 2013 10:30AM - 12:40PM –

Session H3 Multiphase Flows V 325 - Ryan Houim, University of Maryland

10:30AM H3.00001 A Robust Numerical Method for Compressible Dense Granular Flows , RYAN HOUIM, ELAINE ORAN, Department of Aerospace Engineering, University of Maryland — Dense granular flows are important for problems such as coal mine explosions or interior ballistics in which flow compressibility and the presence of shocks are important. Numerical solutions of such flows has been plagued with difficulties arising from non-conservative nozzling terms, which are often neglected for numerical convenience. The “cure” has been to use highly dissipative numerical methods to avoid instability when the non-conservative terms physically must be included. Second-order methods or even refining the grid can reintroduce these numerical instabilities. We developed a robust and high-order numerical method for solving dense granular flows in highly compressible situations that circumvents these problems. The technique has been verified on a number of test problems including advection of a granular material interface, granular shocks, and transmission angles of oblique compaction waves. The method has been demonstrated in challenging situations where a shock impacts a dense layer of dust on very fine meshes approaching the continuum limit of the granular phase.

10:43AM H3.00002 Study of Influence of Experimental Technique on Measured Particle Velocity Distributions in Fluidized Bed , BALAJI GOPALAN, West Virginia University Research Corporation / National Energy Technology Laboratory, FRANK SHAFFER, National Energy Technology Laboratory — Fluid flows that are loaded with high concentration of solid particles are common in oil and chemical processing industries. However, the opaque nature of the flow fields and the complex nature of the flow have hampered the experimental and computational study of these processes. This has led to the development of a number of customized experimental techniques for high concentration particle flows for evaluation and improvement of CFD models. This includes techniques that track few individual particles, measures average particle velocity over a small sample volume and those over a large sample volume. In this work novel high speed PIV (HsPIV), with individual particle tracking, was utilized to measure velocities of individual particles in gas-particle flow fields at the walls circulating and bubbling fluidized bed. The HsPIV measurement technique has the ability to simultaneously recognize and track thousands of individual particles in flows of high particle concentration. To determine the effect of the size of the sample volume on particle velocity measurements, the PDF of Lagrangian particle velocity was compared with the PDF of Eulerian for different domain sizes over a range of flow conditions. The results will show that measured particle velocity distribution can vary from technique to technique and this bias has to be accounted in comparison with CFD simulations.

10:56AM H3.00003 A novel finite element framework for numerical simulation of fluidization processes and multiphase granular flow[1] , JAMES PERCIVAL, ZHIHUA XIE, DIMITRIOS PAVLIDIS, Imperial College London, JEFFERSON GOMES, University of Aberdeen, CHRISTOPHER PAIN, OMAR MATAR, Imperial College London — We present results from a new formulation of a numerical model for direct simulation of bed fluidization and multiphase granular flow. The model is based on a consistent application of continuous-discontinuous mixed control volume finite element methods applied to fully unstructured meshes. The unstructured mesh framework allows for both a mesh adaptive capability, modifying the computational geometry in order to bound the error in the numerical solution while maximizing computational efficiency, and a simple scripting interface embedded in the model which allows fast prototyping of correlation models and parameterizations in intercomparison experiments. The model is applied to standard test problems for fluidized beds.

[1]EPSRC Programme Grant EP/K003976/1

11:09AM H3.00004 Material Point Method and Multi-velocity Formulation for History Dependent Phase Transitions[1] , DUAN ZHANG, XIA MA, Los Alamos National Laboratory — Phase transition has been used to describe both physical and chemical phenomena ranging from melting of ice, erosion of dirt and sand by storm water, combustion of fuel and pulverization of materials. Many of these processes involve effect of history. Tracing history in cases of extreme material deformation has been a significant issue for many numerical methods, especially in cases of phase transitions. The material point method (MPM) is a numerical method based on the wake solution principle of solving a set of partial differential equations. The starting point in this talk is a set of multi-velocity equations obtained based on statistical description of the materials. The equations and the numerical method allow for the use of realistic material models in cases of extreme deformation. The advantage of this statistical description is its convenience in the consideration of the transition between interacting continua to disperse debris flows. MPM uses Eulerian velocity field to describe the extreme material deformation, while uses Lagrangian material points to track material deformation history. We show comparisons of this multi-velocity formulation with traditional approaches and new capabilities of this formation and the numerical method.

[1]Work performed under the auspices of the United States Department of Energy.

11:22AM H3.00005 Computational model and simulations of gas-liquid-solid three-phase interactions[1] , LUCY ZHANG, CHU WANG, Rensselaer Polytechnic Institute — A computational technique to model three-phase (gas-liquid-solid) interactions is proposed in this study. This numerical algorithm couples a connectivity-free front-tracking method that treats gas-liquid multi-fluid interface to the immersed finite element method that treats fully-coupled fluid-solid interactions. The numerical framework is based on a non-boundary-fitted meshing technique where the background grid is fixed where no mesh-updating or re-meshing is required. An indicator function is used to identify the gas from the liquid, and the fluid (gas or liquid) from the solid. Several 2-D and 3-D validation cases are demonstrated to show the accuracy and the robustness of the method.

[1]Funding from NRC and CCNI computational facility at Rensselaer Polytechnic Institute are greatly acknowledged.

11:35AM H3.00006 Analysis on the formation and growth of condensing aerosol particles in a turbulent mixing layer , KUN ZHOU, ANTONIO ATTILI, AMJAD AL-SHAARAWI , FABRIZIO BISETTI, CCRC, KAUST — A simulation of the formation and growth of dibutyl phthalate (DBP) particles in a three-dimensional turbulent mixing layer is performed to investigate the effects of turbulence on the aerosol evolution. A fast, hot stream with DBP vapor is mixed with a slow, cold stream achieving supersaturation by turbulent mixing. The aerosol dynamics are solved with the quadrature method of moments, and the moments are transported via a Lagrangian particles scheme. The results show that aerosol particles are formed in the cold stream, while they grow rapidly in the hot stream. The differential diffusion of temperature/vapor concentration and aerosol particles is investigated through conditional statistics in the mixture fraction space. Aerosol particles formed in the cold stream tend to drift towards the hot stream and grow substantially there.

11:48AM H3.00007 Time resolved measurements of rigid fiber dispersion in near homogeneous isotropic turbulence , LILACH SABBAN, ASAF COHEN, RENE VAN HOUT, Technion - Israel Institute of Technology, EMPFL ENVIRONMENTAL MULTI- PHASE FLOW LABORATORY TEAM — Time resolved, planar particle image velocimetry (PIV, 3kHz) and two-orthogonal view, digital holographic cinematography (2kHz) was used to measure 3D fiber trajectories/orientation dynamics in near homogeneous isotropic air turbulence (HIT) with dilute suspended fibers. The PIV covered a field of view of 6x12 mm^2 and the holography a volume of interest of 17^3 mm^3, positioned at the center of the chamber. HIT (Re_λ =144) was generated in the center of a 40^3 cm^3 cube by eight woofers mounted on each of its corners. Three different nylon fibers having a length of 0.5 mm and diameter of 10, 14 and 19μm were released from the top of the chamber. Fibers had Stokes numbers of order one and are expected to accumulate in regions of low vorticity and settle along a path of local minimal drag. Fiber 3D trajectories/orientations have been obtained from the holography measurements and orientational/translational dispersion coefficients will be presented. In addition the flow field in the vicinity of tracked fibers has been resolved by the PIV, and results on fluid and fiber accelerations and position correlation with in-plane strain rate and out-of-plane vorticity will be presented.

12:01PM H3.00008 ABSTRACT WITHDRAWN –

12:14PM H3.00009 Fine Structure in Energy Dissipation at the onset of turbulence under oscillatory Flow , RUMA DUTTA, University of Southern Mississippi, S. SAJJADI, Embry-Riddle Aeronautical University, FLUID DYNAMICS TEAM, MATHEMATICS TEAM — Fine structure formation studies have been an active area of reserach in recent past and is very much associated in turbulence phenomena. The large scale structures are of ow width and contain most of the energy which dominates transport of mass, momentum and heat. The small scales include dissipative range responsible for most of the energy dissipation and inertial range. Since small scales contain most of interesting formation of structures in terms of physics and simulation that is the most obvious reason we are interested in fine structure formation of the small scale turbulence structure. The analytical side of studies focus mostly on the singularities of Navier Stokes equation have natural connection to far away dissipation range. Small scale turbulence is a fertile ground for studying on vortex breakdown and reconnection that carries many interesting physics. Research on small scale turbulence have remarkable influence heat transfer and particle/chemical phenomena. In this work, we intend to focus on numerical investigation of dynamics of vortex structure of small scale at the onset of turbulence at various Reynold number.

12:27PM H3.00010 Development of iterative algorithms of increased convergence and accuracy for multiphase flow simulation , MAXIM FILATOV, Lomonosov Moscow State University, DMITRY MAKSIMOV, Keldysh Institute of Applied Mathematics — Newton's method is commonly used in reservoir simulation problems. However, it doesn't have the property of globally convergence (Younis et al., 2008). Most of the convergence problems are generally related to change of the nonlinear equation being solved during iteration process (Maksimov et al., 2010): 1) change of phase flow direction; 2) change of the well working target (fixed rate - limit bottom hole pressure - shut); 3) appearance and disappearance of a phase; 4) appearance of movable phase; 5) others, connected with problem formulation improvement. Note that the form of the approximating equations is unknown in advance and is determined by the solution itself. We have considered approaches of improving convergence of Newton's method for reservoir flow problems in general problem formulation, characterized by dependence of the equation approximation form on the solution itself and thus changing of the approximating equation during the iteration process. The approach is based on the universal principle of decreasing of each residual component in all cells. For determination of chopping level, points of approximation form change are employed. As an addition to basic approach, controlled violation of approximation rules was considered, not affecting material balance of system.

Monday, November 25, 2013 10:30AM - 12:14PM –

Session H4 Separated Flows II - Wakes and Flows past Special Surfaces 326 - Gary Coleman, NASA Langley

10:30AM H4.00001 Low-Reynolds number compressible flow around a triangular airfoil , PHILLIP MUNDAY, KUNIHIKO TAIRA, Florida State University, TETSUYA SUWA[1], DAIJU NUMATA, KEISUKE ASAI, Tohoku University — We report on the combined numerical and experimental effort to analyze the nonlinear aerodynamics of a triangular airfoil in low-Reynolds number compressible flow that is representative of wings on future Martian air vehicles. The flow field around this airfoil is examined for a wide range of angles of attack and Mach numbers with three-dimensional direct numerical simulations at $Re = 3000$. Companion experiments are conducted in a unique Martian wind tunnel that is placed in a vacuum chamber to simulate the Martian atmosphere. Computational findings are compared with pressure sensitive paint and direct force measurements and are found to be in agreement. The separated flow from the leading edge is found to form a large leading-edge vortex that sits directly above the apex of the airfoil and provides enhanced lift at post stall angles of attack. For higher subsonic flows, the vortical structures elongate in the streamwise direction resulting in reduced lift enhancement. We also observe that the onset of spanwise instability for higher angles of attack is delayed at lower Mach numbers.

[1]Currently at Mitsubishi Heavy Industries, Ltd., Nagasaki

10:43AM H4.00002 Viscous-Inviscid Interaction Analysis in High-Reynolds Number Flows Using Complex Singularities , K.W. CASSEL, Illinois Institute of Technology, F. GARGANO, M. SAMMARTINO, V. SCIACCA, University of Palermo — Interaction between the viscous boundary layer and inviscid outer flow occurs during unsteady separation on two distinct spatial scales depending upon the Reynolds number regime. Using the impulsively-started flow about a circular cylinder, it is illustrated how these regimes can be identified by tracking singularities in the complex plane based on numerical solutions of the unsteady Navier-Stokes equations. Such an analysis also allows for clear identification in Navier-Stokes solutions of the precursor to the van Dommelen singularity, which occurs in the classical non-interactive boundary-layer equations. The first interaction developing in the separation process is large-scale interaction that is visible for all the Reynolds numbers considered, and it signals the first relevant differences between the boundary-layer and Navier-Stokes solutions. For $Re \geq O(10^4)$, a small-scale interaction corresponding to the van Dommelen singularity follows the large-scale interaction.

10:56AM H4.00003 Investigation of Modal Excitation of a Flexible Cylinder on Vortex Induced Vibrations , ERSEGUN D. GEDIKLI, HARRISON ZIMMER, JASON M. DAHL, Department of Ocean Engineering/ University of Rhode Island — The Vortex-Induced Vibration (VIV) of low mode number flexible cylinders is investigated to observe the effect of modal excitation on synchronization of the wake in a uniform flow. Previous studies have focused on the analysis of two degree of freedom vibrations of a rigid cylinder by controlling the ratio between in-line and cross-flow natural frequencies. The present study holds this natural frequency relation constant while varying the anticipated mode shape associated with structural natural frequencies in air. It is found that a combination of an odd mode shape excited in the cross-flow direction with an even mode shape in the in-line direction results in an incompatible synchronization condition, where the dominant forcing frequency in-line may experience a frequency equal to the cross-flow forcing frequency, a condition not typically observed in rigid cylinder experiments. Excitations of odd mode shapes in both in-line and cross-flow directions result in typical VIV excitation of the flexible body, which compares well with rigid cylinder experiments. Observed motions of the flexible body are forced using a rigid cylinder to visualization the wake.

11:09AM H4.00004 Dynamic mode decomposition of supersonic and transonic wakes of generic space launcher configurations , VLADIMIR STATNIKOV, Institute of Aerodynamics and Chair of Fluid Mechanics, RWTH Aachen University, TARANEH SAYADI, Laboratoire d'Hydrodynamique, Ecole Polytechnique, France, MATTHIAS MEINKE, WOLFGANG SCHROEDER, Institute of Aerodynamics and Chair of Fluid Mechanics, RWTH Aachen University, PETER SCHMID, Laboratoire d'Hydrodynamique, Ecole Polytechnique, France — Dynamic mode decomposition (DMD) is applied to supersonic and transonic wakes of generic space launcher configurations of Mach numbers 0.7 and 6 computed using a zonal RANS/LES approach. The axisymmetric geometry includes a backward facing step that causes the flow to separate. In addition to the separation bubble, acoustic waves are also radiated from the downstream region of the flow. Experimental and numerical observations clearly demonstrate the existence of peaks in the pressure spectra which can be attributed to both the flow inside the wake and the acoustic waves in the freestream. The objective of this work is to apply DMD to the set of numerical data in order to firstly, extract the spatial shape of the modes and secondly, identify their respective frequencies. This allows the dynamics associated to the separation bubble and those of the acoustic waves to be differentiated properly. In addition, with the help of DMD the modes responsible for pressure-loading on the backward face of the step are extracted and analyzed.

11:22AM H4.00005 On the relationship between boundary-layer thickness, base drag and near-wake flow of an axisymmetric bluff body , MARIA VITTORIA SALVETTI, ALESSANDRO MARIOTTI, GUIDO BURESTI, DICI, University of Pisa — A large contribution to the aerodynamic drag of a bluff body is given by the low pressures on its base, i.e. the surface lying within the separated wake. In order to develop strategies to decrease drag, a critical issue is the relationship between the base pressure and the flow parameters. The main application is the development of methods for the reduction of the drag of road vehicles. We consider the flow around an axisymmetric body which can be viewed as a simplified model of a road vehicle, for which experiments, LES and DNS were carried out. Both experiments and simulations showed that an increase of the boundary layer thickness before separation reduces the pressure drag of the body. This is connected with an increase of the length of the mean recirculation region behind the body. A detailed analysis of the near wake dynamics is carried out to ascertain whether the variations of the recirculation length, and hence of the base pressure, caused by the modifications of the boundary layer thickness, may be connected with changes in the dynamics of the vorticity structures originating from the instability of the separated shear layers. More generally, this analysis may be also a useful step towards devising further strategies for pressure drag reduction.

11:35AM H4.00006 Turbulent flow past an obstacle embedded in a hydraulically rough and porous bed[1] , NIKOLAOS APSILIDIS, Virginia Tech, PANAYIOTIS DIPLAS, Lehigh University, CLINTON DANCEY, PAVLOS VLACHOS, POLYDEFKIS BOURATSIS, Virginia Tech — The pressure gradients imposed by wall-mounted obstacles give rise to large-scale, coherent flow structures. Past studies have linked the unsteadiness of these organized flow motions to phenomena such as increased turbulence intensities, momentum and heat transfer over the junction region [Simpson, *Annu. Rev. Fluid Mech.* 2001]. The typical configuration under study refers to a bluff body mounted vertically on an impermeable and hydraulically smooth wall. These characteristics of the bottom boundary, however, do not model accurately a number of flows of environmental (river flow around a boulder) or engineering interest (flow past a bridge pier). Motivated by this inconsistency, we carried out experiments for the turbulent flow at the upstream junction of a cylinder placed within a permeable, hydraulically rough bed. Using 2D2C Particle Image Velocimetry, we captured the rich dynamics of the coherent flow structures developing over the region of interest. We compared results with those from a benchmark test run at a similar Reynolds number, but over a smooth and impermeable bed. Comparisons show that the unsteadiness of coherent flow structures at a wall-body junction increases significantly in the presence of a permeable and rough wall.

[1]This work was supported by the National Science Foundation (EAR 0738759) and the Research Office of the U.S. Army Corps of Engineers (ARO 53512-EV).

11:48AM H4.00007 Flow around a superhydrophobic cylinder[1] , JESSICA SHANG, BRIAN ROSENBERG, PETER DEWEY, HOWARD STONE, Princeton University, ALEXANDER SMITS, Princeton University, Monash University — The boundary condition on a circular cylinder is varied through the use of superhydrophobic surfaces, which introduce a partial-slip boundary condition. We examine the effect of these surfaces on the separation behind a cylinder in the Reynolds number range $15 < Re_D < 1600$. Two different superhydrophobic surfaces are compared with a smooth untreated surface: a conventional superhydrophobic surface consisting of a aluminum hydroxide networked nanostructure with an air-water interface, and a slippery liquid-infused surface (SLIPS) that is similarly nanostructured. We find no change in the critical Re_D or the vortex shedding frequency. An increase in the vortex formation length, generally associated with a decrease in base suction, occurs for the air-water interface for $300 < Re_D < 900$, but not for SLIPS. Superhydrophobic surfaces appear to have no similar effect at higher Re_D in the shear layer transition regime.

[1]ONR N00014-12-1-0875; ONR N00014-12-1-0962

12:01PM H4.00008 Flow over Barnacles–Characterization of Barnacle Geometry and Some Initial Flow Characteristics[1] , JASIM SADIQUE, XIANG YANG, CHARLES MENEVEAU, Johns Hopkins University, MICHAEL SCHULTZ, United States Naval Academy, RAJAT MITTAL, Johns Hopkins University — Macrobiofouling is a serious concern for the marine industry, costing billions in preventive and control measures. Accurate modelling of flows over surfaces with such complex geometry and wide range of length scales is still a huge challenge. Such simulations are required in predicting the effects of fouling, like surface drag and also forces experienced by individual barnacles. DNS or wall resolved LES are impractical due to constraints imposed by the nature of the geometry. We aim to develop and test a computational tool for accurate simulation of such flows. The method being proposed incorporates generalized dynamic wall models along with sharp-interface Immersed Boundary Methods. The results from these simulations will help us understand the effects on surface drag caused by variations in parameters like roughness density, roughness heights, spatial heterogeneity etc. Along with this, detailed studies on a single barnacle will help us in understanding flow structures in the presence of boundary layers. In this talk we will give a brief overview of the problem and some results from our investigation on the characterization of Barnacle geometries and on the characteristics of flow over a single barnacle.

[1]This research is supported by a grant from the Office of Naval Research.

Monday, November 25, 2013 10:30AM - 12:27PM –

Session H5 CFD V 327 - Tomasz Drozda, NASA

10:30AM H5.00001 Simulation of Reacting Flow with a Discontinuous Spectral Element Method , ZIA GHIASI, FARZAD MASHAYEK, JONATHAN KOMPERDA, University of Illinois at Chicago — While using high order methods is desirable in order to accurately capture the small scale mixing effects in reacting flows, the challenge is to develop and implement such methods for complex geometries. In this work, a high-order Discontinuous Spectral Element Method (DSEM) code, which solves for the Navier-Stokes equations, has been modified by adding the appropriate components to solve for scalar transport equations in order to simulate the chemical reaction. Dealing with discontinuous solution at element interfaces is a challenge that is met by patching the fluxes at mortars thus making them continuous on interfaces. The patching is performed using the Lax-Fredrichs numerical flux for scalars, whereas a generalized Riemann solver is used for the Navier-Stokes equations. Direct numerical simulation is conducted in a temporally developing mixing layer to validate the method for a single step reaction ($F + rO \rightarrow [1 + r]P$). Next, the method is implemented to simulate a subsonic reacting flow in a slanted cavity combustor with gaseous fuel injectors to demonstrate the capability of the method to handle complex geometries. The results will be used for physical understanding of mixing and reaction in this type of combustors.

10:43AM H5.00002 A numerical investigation of the influence of aspect ratio in three dimensional separated flows , NIKOLAOS MALAMATARIS, George Mason University/TEI of W.Macedonia — The influence of aspect ratio in three dimensional separated flows is investigated numerically by solving the full three dimensional Navier Stokes equations for Newtonian fluids using standard Galerkin finite elements. As a prototype flow, the backward facing step is chosen with an expansion ratio of 1:2. The Reynolds number is of the order of 1000 where steady state, laminar flow conditions prevail. The computational domain is designed as an actual laboratory experiment with lateral walls and aspect ratios from 1:10 up to 1:40. The results focus on the spanwise variation of the length and the strength of both eddies for this flow that appear along the bottom and top wall. Depending on what attributes of the flow are taken for comparison, aspect ratios of 1:20 up to 1:40 are considered adequate for calling two dimensional flow conditions along the plane of symmetry. The results are contrary to the common wisdom in this field where the aspect ratio of 1:10 is still considered satisfactory two dimensional flow conditions. This is the first computational study for separated flows that raises the issue of two dimensionality along the plane of symmetry and computes the eddy along the top wall for aspect ratios less than 1:35.

10:56AM H5.00003 Ablation patterns driven by simple flows[1] , RYAN CROCKER, DANIEL HAGAN, University of Vermont, MICHAEL ALLARD, University of New Hampshire, YVES DUBIEF, University of Vermont, CHRISTOPHER WHITE, University of New Hampshire — The erosion (here through thermal ablation) of a surface driven by a turbulent, or at least nonlinear, flow may offer an interesting variety of erosion patterns. The present work is interested in the interactions between the flow coherent structures and the topology and erosion rates of such structures. The investigation involves different flows including natural convection flow and flows parallel and perpendicular to the ablated surface. The simulation algorithm is based on momentum and thermal immersed boundary techniques in a finite volume direct numerical simulation flow solver. The interface is tracked by a level set method and the ablation velocity is governed by the Stefan condition. The analysis focuses on the non-equilibrium nature of the flow and the possible prediction of erosion rates.

[1]This work is supported by NSF CBET 0967857 and NASA NNX11AM07A

11:09AM H5.00004 Rich 3-tori dynamics in small-aspect-ratio highly counter-rotating Taylor-Couette flow – reversal of spiraling vortices , SEBASTIAN ALTMEYER, BJÖRN HOF, Institute of Science and Technology Austria, 3400 Klosterneuburg, Austria, FRANCISCO MARQUES, Department of Fisca Aplicada, Universitat Politecnica de Catalanya Girona s/n, Modul B4 Campus Nord, 08034 Barcelona, Spain, JUAN M. LOPEZ, School of Mathematical and Statistical Sciences, Arizona State University, Tempe, Arizona 85287, USA — We present numerical simulations concerning the reversal of spiraling vortices in short highly counter-rotating cylinders. Increasing the differential cylinder rotation results in global flow-inversion which develops various different and complex flow dynamics of several quasi-periodic solutions that differ in their number of vortex cells in the bulk. The dynamics change from being dominated of the inner cylinder boundary layer to be dominated by the outer cylinder boundary layer. Solutions exist on either two or three tori invariant manifolds whereby they appear as symmetric or asymmetric states. We find for either moderate and high inner cylinder rotation speed the quasi-periodic flow to consist of only two vortex cells but differ in its spiraling direction. These both flows live on 2-tori but differ in number of symmetries. While for the quasi-periodic flow at lower rotation speed a pair of symmetrically related 2-tori exists the quasi-periodic flow at higher rotation speeds is symmetric living on a single 2-torus. In addition these both flows differ due to their dominant azimuthal m modes. The 2-tori states are separated by an further quasi-periodic flow living on pair of symmetrically related 3-tori and offer periodical competition between a two and three vortex cell states.

11:22AM H5.00005 Numerical Study of Flow Structure of the Taconis Oscillations in an Axisymmetric Closed Tube , KATSUYA ISHII, ITC, Nagoya Univ., SHYUN KITAGAWA, CSE, Nagoya Univ., SHIZUKO ADACHI, Tokyo International Univ. — Spontaneous thermoacoustic oscillations of a helium gas in a closed cylindrical tube are studied by solving the axisymmetric compressible Navier-Stokes equations. The wall temperature of the hot part near both ends (300K) and that of the cold central part (20K) are fixed. The computations are done for various values of the length ratio of the hot part to the cold part between 0.3 and 1.0. The oscillation states are divided into three groups according to the magnitude of the pressure amplitude, which are the fundamental mode and the second mode of a standing wave, and the oscillation with a shock wave. The states in each group have distinguished features of the vortical flow field. We analyze the effect of vortices on the structure of the temperature distribution and the flow of energy fluxes to gain a better understanding of the mechanism of the thermoacoustic oscillations.

11:35AM H5.00006 ABSTRACT WITHDRAWN –

11:48AM H5.00007 CFD methodology of a model quadrotor , BURAK SUNAN, Bahcesehir University — This paper presents an analysis of the aerodynamics characteristics of a quadrotor for both steady and unsteady flows. For steady flow cases, aerodynamics behaviour can be defined readily for any aerial vehicles in wind tunnels. However, unsteady flow conditions in wind tunnels make experimental aerodynamics characterizations difficult. This article describes determination of lift, drag and thrust forces on a model quadrotor by using CFD (Computational Fluid Dynamics) software ANSYS Fluent. A significant issue is to find a new CFD methodology for comparison with the experimental results. After getting sufficiently close agreement with some benchmarking experiments, the CFD methodology can be performed for more complicated geometries. In this paper, propeller performance database experiments from Ref. 1 will be used for validation of the CFD procedure. The results of the study reveals the dynamics characteristics of a quadrotor. This demonstrates feasibility of designing a quadrotor by CFD which saves time and cost compared to experiments.

12:01PM H5.00008 Using Navier-Stokes to Characterize Re-Entry of Microscale Vehicles , SUDHARSAN THIRUVENKADAM, HARRIS BEN, The University of Texas at Arlington, Arlington, Texas — Atmospheric reentry vehicles experience different flow regimes during flight due to the change in atmospheric density. This change in density creates non-equilibrium regions on the order of one mean free path, called as Knudsen layer. In the design of atmospheric reentry vehicles, the flux variations near solid surface are of critical importance. The traditional CFD simulations which use Navier Stokes equations fail to predict the flow in Knudsen layer. These areas where the rarefaction effects begin to dominate can be quantified by the Knudsen breakdown parameter. The Direct Simulation Monte Carlo (DSMC) method, although accurate for all flow regimes, it is computationally expensive as the number of simulating molecules increases. We developed a method that models the Knudsen Layer by using Navier Stokes equations with Maxwell-Smoluchowski slip boundary conditions and DSMC for low (Kn $<$ 0.1) and high (Kn $>$ 0.1) Knudsen numbers respectively. This study investigates the surface properties of a flat plate with Nitrogen gas flow from continuum to rarefied regimes. Computational fluid dynamics and DSMC results are obtained for different test conditions. The results demonstrate that the Knudsen layer can be predicted with DSMC and continuum approach for all flow regimes.

12:14PM H5.00009 von Neumann Stability Analysis of Pressure-Based Formulation of 1D and 2D Euler Equations , SANTOSH KONANGI, URMILA GHIA, University of Cincinnati — The stability properties of a pressure-based scheme for the Euler equations are investigated, as such schemes are widely employed in commercial computational fluid dynamics codes. The published literature often focusses on model equations, and does not consider the solution scheme used in the parent code. The present study conducts a von Neumann stability analysis for a pressure-based, segregated scheme, SIMPLE (Semi-Implicit Method for Pressure-Linked Equations). The 1D and 2D Euler equations, closed by an "artificial" equation of state, are discretized using finite differences on a staggered grid, which permits equivalence to finite-volume discretization. As a first effort, first-order accurate spatial and temporal schemes are analyzed, to determine error amplification matrices, identify stable and unstable regimes, and predict practical stability limits in terms of the maximum allowable CFL number as a function of Mach number. The predictions are verified using the Riemann problem at several Mach numbers, and very good agreement is obtained between the predicted and the "numerically" observed CFL values. Hence, the present results should prove useful in guiding the stability of a simulation using the parent-code and the scheme tested here.

Monday, November 25, 2013 10:30AM - 12:14PM –

Session H6 Microfluids: Fluidic Devices I 328 - Thomas Cubaud, Stony Brook University

10:30AM H6.00001 Foam imbibition in a Hele-Shaw cell via laminated microfluidic "T-junction" device , DINA PARRA, THOMAS WARD, Iowa State University — In this talk we analyze experimental results of a novel microfluidic "T-junction" device, made from laminated plastic, that is used to produce foam in porous media. The fluids, both Newtonian and non-Newtonian liquids and air, are driven using constant-static pressure fluid pumping. For the T-junction geometry studied there are novel observations with this type of pumping: 1) at low pressure ratios there is an increase in the liquid and total flow rates and 2) at higher pressure ratios there is a decrease in the liquid flow rate. To understand this phenomenon we visualize the drop production process near the T-junction. Furthermore, flow rates for the liquid and total volume are estimated by imbibing the foam into a Hele-Shaw cell. Foam is produced by using a mixture containing aqueous polyacrylamide of concentrations ranging from 0.01-0.10% by weight and several solution also containing a sodium-lauryl-sulfate (SLS) surfactant at concentrations ranging 0.01-0.1% by weight.

10:43AM H6.00002 Gas rivulets on a submerged solid surface: a new microfluidic technique to produce microbubbles , MIGUEL A. HERRADA, ALFONSO M. GAÑÁN-CLAVO, University of Seville, JOSÉ M. MONTANERO, University of Extremadura — A general microfluidic technique is proposed to produce microbubbles from gas micro-rivulets formed on the surface of a solid over which a liquid is flowing. In the particular geometry considered in this work, a gaseous stream is injected through a T-junction into a channel transporting a liquid current. The gas adheres to a hydrophobic strip printed on the channel surface. When the gas and liquid flow rates are set appropriately, a gaseous rivulet flows over that strip. The rivulet breaks up downstream due to a capillary pearling instability, which leads to a monodisperse collection of microbubbles. The physics of the process is described from both the numerical simulation of the Navier-Stokes equations, and the linear stability analysis of an infinite gaseous rivulet driven by a coflowing liquid stream. This analysis allows one to determine a necessary condition to get this effect in a T-junction device. It also provides reasonably good predictions for the size of the produced microbubbles.

10:56AM H6.00003 Utilizing chemo-mechanically functionalized oscillating fins to "catch and release" nanoparticles in binary flow , YA LIU, University of Pittsburgh, YONGTING MA, Virginia Commonwealth University, AMITABH BHATTACHARYA, Indian Institute of Technology Bombay, OLGA KUKSENOK, University of Pittsburgh, XIMIN HE, JOANNA AIZENBERG, Harvard University, ANNA BALAZS, University of Pittsburgh — In biomimetics, designing an effective "catch and release" device for the selective removal of target species from the surrounding solution is critical for developing autonomous sensors and sorters. Using computational simulation, we model an array of oscillating fins that are tethered on the floor of a microchannel and immersed in a binary-fluid stream. During the oscillation, the fins with the specific chemical wetting reach the upper fluid when they are upright and are entirely within the lower stream when they are tilted. We introduce specific adhesive interactions between the fins and particulates in the solution and determine conditions where the oscillating fins can selectively bind ("catch") target nanoparticles within the upper fluid stream and then release these particles into the lower stream. We isolate the effects of chemical wetting on the fins (e.g., wetting contact angle between fins and fluid) and mechanical parameters (e.g., frequency of fins' oscillations) that lead to the efficient extraction of target species from the upper stream and placement into the lower fluid. Our understanding provides fundamental insights into the system's complex dynamics and mechanism for detection, separation, and purification of multi-component mixtures.

11:09AM H6.00004 Transporting Janus Nanoparticles Using Self-Healing Vesicles , XIN YONG, EMILY CRABB, NICHOLAS MOELLERS, ISAAC SALIB, GERALD MCFARLIN, OLGA KUKSENOK, ANNA BALAZS, University of Pittsburgh — Using dissipative particle dynamics (DPD) simulations, we model the interaction between nanoscopic lipid vesicles and Janus nanoparticles in the presence of an imposed flow. Both the vesicle and Janus nanoparticles are localized on a hydrophilic substrate and immersed in a hydrophilic solution. The fluid-driven vesicle successfully picks up Janus particles on the substrate and transports these particles as cargo along the surface. With the introduction of a "sticky" domain or a nanoscale crack onto the otherwise flat substrate, the vesicles can robustly drop off and deposit the particles at the targeted places. For Janus particles with a large hydrophobic region, the vesicle tears and deposits the particle with a few lipids covering its hydrophobic region. This lipids coating can protect the particle from the outer solution after deposition. The vesicle then heals itself after tearing off the lipids, and could be reused for transporting particles. These environmentally adaptive and self-healing vesicles can play an important role in drug-delivery and microfluidic applications.

11:22AM H6.00005 Life after wetting: Transport and concentration in paper-based microfluidics using ion concentration polarization , BRENDAN MACDONALD, University of Ontario Institute of Technology, MAX GONG, PEI ZHANG, DAVID SINTON, University of Toronto — In this talk, we present a method for active transport and concentration in paper-based microfluidic devices using ion concentration polarization. Paper-based devices rely on wicking for fluid transport and therefore have limited transport capacity upon reaching a wetted state. We present two methods, one external, and one embedded within the paper to enable transport after wetting is complete. The external device contains a nano-porous membrane and electrical connections required for ion concentration polarization. The device can be placed against the paper surface for transport and concentration. The imbedded method involves patterning the nano-porous membrane within the paper layer and electrical connections in contact with the wetted paper. We demonstrate transport and concentration in paper-based devices using both methods for dyes (fluorescent and non fluorescent), and for biological analytes in a lateral flow device.

11:35AM H6.00006 Buckling of Dielectric Elastomeric Plates for Electrically Active Microfluidic Pumps , DOUGLAS HOLMES, BEHROUZ TAVAKOL, Virginia Tech, MICHAEL BOZLAR, GUILLAUME FROEHLICHER, HOWARD STONE, ILHAN AKSAY, Princeton University — Fluid flow can be directed and controlled by a variety of mechanisms within industrial and biological environments. Advances in microfluidic technology have required innovative ways to control fluid flow on a small scale, and the ability to actively control fluid flow within microfluidic devices is crucial for advancements in nanofluidics, biomedical fluidic devices, and digital microfluidics. In this work, we present a means for microfluidic control via the electrical actuation of thin, flexible valves within microfluidic channels. These structures consist of a dielectric elastomer confined between two compliant electrodes that can be actively and reversibly buckle out of plane to pump fluids from an applied voltage. The out-of-plane deformation can be quantified using two parameters: net change in surface area and the shape of deformation. Change in surface area depends on the voltage, while the deformation shape, which significantly affects the flow rate, is a function of voltage, and the pressure and volume of the chambers on each side of the thin plate. The use of solid electrodes enables a robust and reversible pumping mechanism that will have will enable advancements in rapid microfluidic diagnostics, adaptive materials, and artificial muscles.

11:48AM H6.00007 Towards 2D field-flow fractionation - Vector separation over slanted open cavities , JORGE A. BERNATE, MENGFEI YANG, HONG ZHAO, Stanford University, SUMEDH RISBUD, COLIN PAUL, MATTHEW DALLAS, KONSTANTINOS KONSTANTOPOULOS, Johns Hopkins University, GERMAN DRAZER, Rutgers University, ERIC S.G. SHAQFEH, Stanford University — Planar microfluidic platforms for vector chromatography, in which different species fan out in different directions and can be continuously sorted, are particularly promising for the high throughput separation of multicomponent mixtures. We carry out a computational study of the vector separation of dilute suspensions of rigid and flexible particles transported by a pressure-driven flow over an array of slanted open cavities. The numerical scheme is based on a Stokes flow boundary integral equation method. The simulations are performed in a periodic system without lateral confinement, relevant to microfluidic devices with negligible recirculation in the main channel. We study the deflection of rigid spherical particles, of flexible capsules as a model of white and red blood cells, and of rigid discoidal particles as a model of platelets. We characterize the deflection of different particles as a function of their size, shape, shear elasticity, their release position, and the geometric parameters of the channel. The simulations provide insight into the separation mechanism and allow the optimization of specific devices depending on the application. Good agreement with experiments is observed.

12:01PM H6.00008 Large modulation of light beams by surface acoustic waves , BYUNG HANG HA, KANG SOO LEE, GHULAM DESTGEER, JIN HO JUNG, HYUNG JIN SUNG, KAIST, FLOW CONTROL LAB TEAM — We present a refraction-based method for light beam deflection with up to 20 degrees deflection angle and modulation bandwidth on the order of ten hertz. The mechanism utilizes sharp focusing of acoustic energy to produce a steep gradient of refractive index inside an optically-transparent media. The medium, fluid or solid, is subject to bulk acoustic waves (leaky Rayleigh waves) which is transmitted from piezo-actuated surface acoustic waves. The wavelets interfere to form a vertical gradient of acoustic energy density in the media as well as a refractive index gradient accordingly. Given the input acoustic energy, the biggest deflection angle is obtained when the light beam is given at the area where the refractive index gradient is largest. The device is based on lens effect and free from numerous limitations that acousto-optic deflector has: precise alignment of the incident angle of light beam is unnecessary, 100% deflection is achieved, the device is modulated by amplitude, not frequency and the deflection efficiency is not dependent on the polarity and the wavelength of light beam. The device can deflect, switch, and scan light beams and is applicable to pre-press, radar, laser imaging and displays, instrumentation and research.

Monday, November 25, 2013 10:30AM - 12:27PM –

Session H7 Microfluids: Interfaces and Wetting II 329 - Lou Kondic, New Jersey Institute of Technology

10:30AM H7.00001 Flow through a thin film on non-flat substrates , CIRO SEMPREBON, Max Plank Institute for Dynamics and Self Organization, MARTIN BRINKMANN, Saarbucken Universty — The ability of liquids to form films on surfaces is essential for many technical applications such as the coating of surfaces or the liquid transport. While for many heterogeneous materials it is possible to introduce effective material properties from spatial averaging, here close to the percolation transition the size of a representative domain becomes comparable to the size of the whole system. In this work we investigate the morphological evolution and the transport properties of a thin wetting layer adhering to an irregular rough substrate. Static film profiles are obtained by numerically minimizing the interfacial energy including a generic short ranged interface potential to account for a precursor film and a finite apparent contact angle. Assuming that the flow does not alter the profile of the liquid meniscus, we employ the static film configurations resulting from the energy minimization to solve the linearized steady thin film equation and obtain the total volume flux. Our results show that the connectivity between liquid domains plays a key role in predicting the transport properties of the liquid interface.

10:43AM H7.00002 Altering the Flow of Gas through Modification of Surface Films , DONGJIN SEO, WILLIAM DUCKER, Virginia Tech — Normally the flow of gas in a channel is considered to be a function of the pressure difference between the ends of the channel and the geometry. For high Knudsen numbers, the flow also depends on the tangential momentum accommodation coefficients (TMAC). Here we consider methods of altering the TMAC, and thus the flow of gas at 1 atm through a narrow channel, by the use of surface films that alter the TMAC. Gas flow was determined by measuring the damping on a glass sphere as a function of separation from a flat plate. The solids were coated with octadecyltrichlorosilane (OTS), which undergoes a melting-like transition near room temperature. The measured damping passes through a maximum in the temperature range of 9 - 42 °C and thus the TMAC also passes through a maximum. We attribute this maximum to competing effects due to the decrease in surface roughness and the decrease in stiffness as a function of temperature. Control of flow via alteration of a surface film should also be possible using other methods of altering the state of surface films.

10:56AM H7.00003 Mass Transfer of Gas on Slippery Superhydrophobic Surface , ELIF KARATAY, PEICHUN AMY TSAI, ROB LAMMERTINK, University of Twente, SOFT MATTER, FLUIDICS AND INTERFACES GROUP TEAM — Superhydrophobic substrates containing gas bubbles are advantageous for generating hydrodynamic slippage. When a viscous liquid flowing upon, bubble surfaces provide shear-free gas-liquid interfaces thereby slippage. Besides, the absorption of gas into the liquid occurs at the bubble surfaces. We experimentally measure and numerically estimate the mass transfer of gas absorption at the stable gas/liquid interfaces for short contacting times. We study the net rate of gas absorption experimentally by in-situ measurements of dissolved oxygen concentration profiles in aqueous solutions flowing over oxygen bubbles using fluorescent lifetime imaging microscopy. We numerically analyze the dynamics of interfacial mass transfer of dissolved oxygen, by considering (i) kinetic equilibrium conditions at bubble surfaces that is conventionally described by Henry's Law and (ii) non-equilibrium conditions at bubble surfaces using Statistical Rate Theory (SRT). Our experimental results show that kinetic equilibrium is not established for short contact times. Mass transfer of gas into liquid flow past micro-bubbles can be well described by our simulations performed with the non-equilibrium theory for short exposure time ($\sim$ 180 μs) of liquid with a microbubble, deviating from the commonly

11:09AM H7.00004 Drag Reduction using Superhydrophobic Sanded Teflon , DONG SONG, ROBERT DANIELLO, JONATHAN ROTHSTEIN, University of Massachusetts - Amherst — In this talk, we present a series of microfluidic experiments designed to investigate drag reduction using series of roughened Teflon surfaces. The Teflon surfaces where made superhydrophobic by imparting surface texture through sanding with sand papers with a range of grit sizes. Our previous work showed that there exists an optimal sand paper grit (240 grit) for eliminating contact angle hysteresis. We will show that a Teflon surface roughened with the same sand paper grit also maximizes the drag reduction and the slip length observed in laminar flows. Increasing or decreasing the grit size was found to reduce the drag reduction and slip length. A number of different sanding protocols were investigated including sanding preferentially in the flow direction, normal to the flow direction and with a randomized circular pattern. Of these three techniques, sanding in the flow direction was found to maximize the slip length.

11:22AM H7.00005 Lubricant-impregnated surfaces for drag reduction in viscous laminar flow , BRIAN SOLOMON, KARIM KHALIL, KRIPA VARANASI, Massachusetts Institute of Technology, MIT TEAM — For the first time, we explore the potential of lubricant impregnated surfaces (LIS) in reducing drag. LIS, inspired by the surface of the Nepenthes pitcher plant, have been introduced as a novel way of functionalizing a surface. LIS are characterized by extremely low contact angle hysteresis and have been show to effectively repel various liquids including water, oils, ketchup and blood. Motivated by the slippery nature of such surfaces, we explore the potential of LIS to reduce drag in internal flows. We observe a reduction in drag for LIS surfaces in a viscous laminar drag flow and model the impact of relevant system parameters (lubricant viscosity, working fluid viscosity, solid fraction, depth of texture, etc.).

11:35AM H7.00006 Flow-driven failure of liquid-filled surfaces , IAN JACOBI, JASON WEXLER, HOWARD STONE, Princeton University — A micro-patterned surface impregnated with liquid is subjected to laminar shear flow and the subsequent drainage of the liquid is measured. Such lubricant-infused rough surfaces offer a promising new approach to drag reduction by providing surface slip between the mobile lubricant within the roughness and the outer flow in a manner that is more stable and robust than traditional air-layer-based super-hydrophobic surfaces. We consider the effect of roughness geometry and lubricant properties on the drainage behavior of liquid-infused surfaces in order to understand the physical mechanisms of liquid drainage and the failure modes of such drag-reducing surfaces at the micro-scale. The analysis of micro-scale drainage is then used to develop design criteria for enhanced lubricant retention and drag reduction under a variety of shear flow conditions.

11:48AM H7.00007 Molecular dynamics simulations of disjoining pressure effects in ultra-thin water films on a metal surface , HAN HU, YING SUN, Drexel University — Disjoining pressure, the excess pressure in an ultra-thin liquid film as a result of van der Waals interactions, is important in lubrication, wetting, flow boiling, and thin film evaporation. The classic theory of disjoining pressure is developed for simple monoatomic liquids. However, real world applications often utilize water, a polar liquid, for which fundamental understanding of disjoining pressure is lacking. In the present study, molecular dynamics (MD) simulations are used to gain insights into the effect of disjoining pressure in a water thin film. Our MD models were firstly validated against Derjaguin's experiments on gold-gold interactions across a water film and then verified against disjoining pressure in an argon thin film using the Lennard-Jones potential. Next, a water thin film adsorbed on a gold surface was simulated to examine the change of vapor pressure with film thickness. The results agree well with the classic theory of disjoining pressure, which implies that the polar nature of water molecules does not play an important role. Finally, the effects of disjoining pressure on thin film evaporation in nanoporous membrane and on bubble nucleation are discussed.

12:01PM H7.00008 Computational modelling of microfluidic capillary breakup phenomena , YUAN LI, JAMES SPRITTLES, OCCAM, Mathematical Institute, University of Oxford, JIM OLIVER, Mathematical Institute, University of Oxford — Capillary breakup phenomena occur in microfluidic flows when liquid volumes divide. The fundamental process of breakup is a key factor in the functioning of a number of microfluidic devices such as 3D-Printers or Lab-on-Chip biomedical technologies. It is well known that the conventional model of breakup is singular as pinch-off is approached, but, despite this, theoretical predictions of the global flow on the millimetre-scale appear to agree well with experimental data, at least until the topological change. However, as one approaches smaller scales, where interfacial effects become more dominant, it is likely that such unphysical singularities will influence the global dynamics of the drop formation process. In this talk we develop a computational framework based on the finite element method capable of resolving diverse spatio-temporal scales for the axisymmetric breakup of a liquid jet, so that the pinch-off dynamics can be accurately captured. As well as the conventional model, we discuss the application of the interface formation model to this problem, which allows the pinch-off to be resolved singularity-free, and has already been shown to produce improved flow predictions for related "singular" capillary flows.

12:14PM H7.00009 Computational and experimental investigation of capillary self-focusing in a microfluidic system , S. AFKHAMI, New Jersey Institute of Technology, M. HEIN, R. SEEMANN, Saarland University, L. KONDIC, New Jersey Institute of Technology — We present a capillary focusing method for generating monodisperse submicrometric droplets. The emulsification technique relies on an abrupt change in the aspect ratio of a single shallow and wide microchannel that merges into a deep reservoir [Appl. Phys. Lett. 88:024106 (2006)]. We present a computational framework, supported by experimental observation, to address the capillary self-focusing, in which the interface between the two fluids takes the shape of a tongue narrowing in the flow direction just ahead of the holding reservoir. Our numerical approach is based on a volume-of-fluid method for computing the interface motion and for modeling the surface tension in a Hele-Shaw flow. We present and compare numerical and experimental results for the width of the tongue and predict and measure the transition between two different emusification mechanisms occuring in this geometry.

Monday, November 25, 2013 10:30AM - 12:01PM –

Session H8 Magnetohydrodynamics I 330 - Eric Edlund, Princeton Plasma Physics Laboratory

10:30AM H8.00001 A new divergence-free-preserving high-order scheme for magnetohydrodynamics[1] , SOSHI KAWAI, Institute of Space and Astronautical Science, Japan Aerospace Exploration Agency — We present a new strategy that is very simple, divergence-free, high-order accurate, yet has an effective discontinuous-capturing capability for simulating magnetohydrodynamics (MHD) with shock waves. The new strategy is to construct artificial diffusion terms in a physically-consistent manner, and to be built into the induction equations in a conservation law form at a partial-differential-equation level. The physically-consistent manner means that the artificial terms act as a diffusion term only in the curl of magnetic field to capture numerical discontinuities in the magnetic field while not affecting the divergence field (thus maintaining divergence-free constraint). The proposed method is inherently divergence-free both ideal and resistive MHD, with and without shock waves, and also both inviscid and viscous flows. The method is based on finite difference method with co-located variable arrangement, and any linear finite difference scheme in an arbitrary order (i.e., any desirable high-order) of accuracy can be used to discretize the modified governing equations to ensures the divergence-free constraint numerically at the discretization level. Two-dimensional smooth and non-smooth ideal MHD problems are considered to show a superior performance of the proposed method.

[1]This study was supported by the International Top Young Fellowship Program at Japan Aerospace Exploration Agency.

10:43AM H8.00002 ABSTRACT WITHDRAWN –

10:56AM H8.00003 Mixed convection in duct flows with very strong transverse magnetic fields[1] , XUAN ZHANG, XINYAN LV, LI LIU, ANDREW SCHIGELONE, OLEG ZIKANOV, University of Michigan - Dearborn — Mixed convection in flows of liquid metals within ducts with one heated wall and imposed transverse magnetic field is studied using high-resolution DNS and linear stability analysis. The main attention is given to the cases of strong heating (the Grashof number up to 10^{12}) and strong magnetic field (the Hartmann number up to 800). Various orientations of the duct, temperature gradient, and magnetic field are studied in our project. This presentation is focused on the configuration of a horizontal duct with bottom heating and horizontal transverse magnetic field. It is found that, while conventional turbulence is suppressed, a new type of convection instability appears at high Hartman numbers. The most unstable modes are the rolls aligned with the magnetic field. Their streamwise wavelength is of the order of the width of the duct and decreases with the Hartmann number as the rolls become localized in the lower part of the duct. In fully developed secondary regimes, transport of the rolls by mean flow leads to strong low-frequency oscillations of local temperature.

[1]Work was financially supported by the U.S. NSF (Grant CBET 1232851)

11:09AM H8.00004 ABSTRACT WITHDRAWN –

11:22AM H8.00005 Simulation of liquid metal duct flow at finite magnetic Reynolds number , VINODH KUMAR BANDARU, THOMAS BOECK, JOERG SCHUMACHER, TU Ilmenau, Germany — Turbulent conducting flows at finite magnetic Reynolds numbers occur in magnetohydrodynamic turbulence in plasmas, and in the generation of magnetic fields by the dynamo effect. In simulations the former case is typically studied as box turbulence without walls, and the latter in a closed spherical fluid domain. We are interested in turbulent liquid-metal duct flows in the presence of an exterior localized magnetic field, which is of interest for metallurgical applications. It can be expected to show complex interactions between the field and the flow, which modify both the field and velocity distribution. The evolution of the perturbation in the imposed magnetic field together with the turbulent velocity field in the duct are solved numerically using finite differences through the coupled system of Navier-Stokes and magnetic field transport equations. Characterizing the continuity of the magnetic field perturbations between the exterior and interior of the domain gives rise to non-local boundary conditions which are dealt with the boundary element method. Details of the methodology for numerical computation will be discussed.

11:35AM H8.00006 Transitional liquid metal duct flow near a magnetic dipole , SASKIA TYMPEL, THOMAS BOECK, JOERG SCHUMACHER, University of Technology Ilmenau, Germany — The flow transformation and the generation of vortex structures by a strong magnetic dipole field in a liquid metal duct flow is studied by means of three-dimensional direct numerical simulations. The dipole is considered as the paradigm for a magnetic obstacle which will deviate the streamlines due to Lorentz forces which act on the fluid elements. The duct is of square cross-section. The dipole is located above the top wall and is centered in spanwise direction. Our model uses the quasi-static approximation which is applicable in the limit of small magnetic Reynolds numbers. The analysis covers the stationary flow regime at small hydrodynamic Reynolds numbers Re as well as the transitional time-dependent regime at higher values which may generate a turbulent flow in the wake of the magnetic obstacle. We present a systematic study of these two basic flow regimes on Re and on the Hartmann number Ha, a measure of the strength of the magnetic dipole field. Furthermore, several orientations and positions of the dipole are compared. The most efficient generation of turbulence at a fixed distance above the duct follows for the spanwise orientation which is caused by a certain configuration of Hartmann layers and reversed flow at the top plate.

11:48AM H8.00007 Flow velocimetry for weakly conducting electrolytes based on high resolution Lorentz force measurement. , CHRISTIAN RESAGK, RESCHAD EBERT, SUREN VASILYAN, ANDREAS WIEDERHOLD, Ilmenau University of Technology — We demonstrate that a flow velocity measurement can be transformed into a non-invasive force measurement by metering the drag force acting on a system of magnets around a flow channel. This method is called Lorentz force velocimetry and has been developed in the last years in our institute. It is a feasible principle for materials with large conductivity like liquid metals. To evolve this method for weakly conducting fluids like salt water or molten glass the drag force measurement is the challenging bottleneck. Here forces of 10^{-8} and less of the weight force of the magnet system have to be resolved. In this paper different force measurement techniques get tested and compared. For the current setup the magnet system is attached to a state of the art electromagnetic force compensation balance. Different ways of getting the correct force signal out of the two measurement setups will be presented and discussed. For generalization of the measurement principle the Lorentz force is determined for different fluid profiles. In addition to that we have developed new systematic noise reduction methods to increase the resolution of the force measurement techniques by a factor of ten or larger which we will present here.

Monday, November 25, 2013 10:30AM - 12:27PM –

Session H9 Instability: Interfacial and Thin-Film IV - Elasticity and Substrates 333 - Yuan-nan Young, New Jersey Institute of Technology

10:30AM H9.00001 Long-wave Dynamics of a Membrane in an Electric Field , YUAN-NAN YOUNG, New Jersey Institute of Technology, SHRAVAN VEERAPANENI, University of Michigan, MICHAEL MIKSIS, Northwestern University — We investigate the long-wave non-linear dynamics of an inextensible capacitive elastic membrane under electric fields. The long-wave formulation allows us to analyze the equilibrium membrane profile in a d.c. field. Numerical studies of the governing equation with an integral constraint (for the constant membrane area) elucidate how the membrane bending modulus, electric potential, and frequency of the a.c. electric field gives rise to various membrane dynamics and equilibrium profiles. Pillar formation under the constant area constraint is found as we decrease the membrane bending modulus or increase the electric potential. Increasing the a.c. field frequency stabilizes the membrane profile, and we find the surrounding fluid flow to correlate closely with the local membrane curvature.

10:43AM H9.00002 The effect of structuring on the stability of electrolyte films , CHRISTIAAN KETELAAR, VLADIMIR AJAEV, Southern Methodist University — We investigate the stability of a thin liquid film of an electrolyte on a structured surface with a periodic array of gas-filled grooves. The electrostatic component of disjoining pressure is incorporated into the system of lubrication-type equations to derive a nonlinear evolution equation for film thickness. Electric charges are present at the liquid-gas interface at the top of the film, as well as the solid-liquid interface segments between the grooves and the menisci separating the gas phase in the grooves from the liquid film. We analyze how the length of the groove, the slip length at the groove menisci, and the electric charges at the interfaces affect the stability of the electrolyte film. In particular, we identify the conditions when a stable electrolyte film on a charged surface becomes destabilized by the effect of the structuring.

10:56AM H9.00003 The interaction between viscous fingering and wrinkling in elastic-walled Hele-Shaw cells , DRAGA PIHLER-PUZOVIC, ANNE JUEL, MATTHIAS HEIL, University of Manchester — The development of viscous fingers in circular Hele-Shaw cells is a classical and widely-studied fluid mechanical problem. The introduction of wall elasticity (via the replacement of one of the bounding plates by an elastic membrane) can weaken or even suppress the fingering instability, but it also makes the system susceptible to additional solid-mechanical instabilities. We show that in elastic-walled Hele-Shaw cells that are bounded by sufficiently thin elastic sheets the (fluid-based) viscous fingering instability can arise concurrently with a (solid-based) wrinkling instability. We study the interaction between these distinct instabilities, using a theoretical model that couples the depth-averaged lubrication equations for the fluid flow to the Föppl–von Kármán equations which describe the deformation of the thin elastic sheet. By using a combination of a linear stability analysis and direct numerical simulations, we show that system's behaviour may be characterised by a non-dimensional parameter that indicates the strength of the fluid-structure interaction. For small [large] values of this parameter the system's behaviour is dominated by viscous fingering [wrinkling], with strong interactions between the two instabilities arising in an intermediate regime.

11:09AM H9.00004 Bending of an elastic cantilever by gravity-driven flow of a liquid film , MARINELA POPOVA, HYOUNGSOO KIM, Princeton University, PETER HOWELL, University of Oxford, HOWARD STONE, Princeton University — We experimentally study a gravity-driven liquid flow on a flexible beam. The elastic material bends due to the weight of the liquid. The relationship between hydrodynamics and elasticity is investigated by varying an applied flow rate, the bending stiffness of the beam, and the beam length. Surface tension effects are negligible for these experiments. We compare our results with a model that predicts the beam deformation in terms of two dimensionless parameters, one representing a dimensionless beam length and the other representing a dimensionless beam stiffness. The results span both small deformations as well as large deformations of the cantilever.

11:22AM H9.00005 Stability Theory for Interfacial Patterns in Magnetic Pulse Welding , ALI NASSIRI, GREGORY CHINI, BRAD KINSEY, University of New Hampshire, UNH TEAM — Magnetic Pulse Welding (MPW) is a solid state, high strain-rate joining process in which a weld of dissimilar or similar materials can be created via high-speed oblique impact of two workpieces. Experiments routinely show the emergence of a distinctive wavy pattern, with a well defined amplitude and wavelength of approximately 20 and 70 micrometers, respectively, at the interface between the two welded materials. Although the origin of the wavy pattern has been the subject of much investigation, a unique fundamental physical theory for this phenomenon is as yet not widely accepted. Some researchers have proposed that the interfacial waves are formed in a process akin to Kelvin-Helmholtz instability, with relative shear movement of the flyer and base plates providing the energy source. Here, we employ a linear stability analysis to investigate whether the wavy pattern could be the signature of a shear-driven high strain-rate instability of an elastic-plastic solid material. Preliminary results confirm that an instability giving rise to a wavy interfacial pattern is possible.

11:35AM H9.00006 Instability-Driven Streak Formation During Convective Deposition , ALEXANDER WELDON, KEDAR JOSHI, JAMES GILCHRIST, Lehigh University — The formation of streaks, often unwanted localized multilayer particle deposition oriented in the direction of deposition, is studied across a robust variety of process conditions. In systems where streaks form, many streaks are initiated and then coarsen and merge as the deposition progresses. Image analysis of streaks give a detailed account of their formation and development. Insights into the formation of this stochastic process will be given and suggest ways to suppress the formation of streaks.

11:48AM H9.00007 Effect of lateral vibrations during convective deposition , TANYAKORN MUANGNAPOH, ALEXANDER WELDON, JAMES GILCHRIST, Lehigh University — Vibration-assisted convective deposition is an advanced technique for improving a convective deposition. By adding a mechanical substrate vibration, drastic alterations of an interfacial liquid surface and an evaporate rate were observed. Bond, capillary and Reynolds dimensionless numbers were investigated for describing a change of an interfacial liquid surface. In this experiment, aqueous binary suspensions of colloidal microspheres and nanoparticles were used for studying effect of the amplitudes (0-250 μm) and the frequencies (1-50 Hz) of substrate vibration. The quality of deposited thin films was characterized by using a confocal laser scanning microscope, a scanning electron microscope and an image analysis. The monolayer structures formed from this rapid process can be used in a variety of optical, chemical, and biochemical sensing applications such as optical microlens arrays, microporous membranes and cell capture substrates.

12:01PM H9.00008 Faraday instability on patterned surfaces , JIE FENG, GREGORY RUBINSTEIN, IAN JACOBI, HOWARD STONE, Princeton University — We show how micro-scale surface patterning can be used to control the onset of the Faraday instability in thin liquid films. It is well known that when a liquid film on a planar substrate is subject to sufficient vibrational accelerations, the free surface destabilizes, exhibiting a family of non-linear standing waves. This instability remains a canonical problem in the study of spontaneous pattern formation, but also has practical uses. For example, the surface waves induced by the Faraday instability have been studied as a means of enhanced damping for mechanical vibrations (Genevaux, et. al. 2009). Also the streaming within the unstable layer has been used as a method for distributing heterogeneous cell cultures on growth medium (Takagi, et al. 2002). In each of these applications, the roughness of the substrate significantly affects the unstable flow field. We consider the effect of patterned substrates on the onset and behavior of the Faraday instability over a range of pattern geometries and feature heights where the liquid layer is thicker than the pattern height. Also, we describe a physical model for the influence of patterned roughness on the destabilization of a liquid layer in order to improve the design of practical systems which exploit the Faraday instability.

12:14PM H9.00009 Bottom reconstruction in power-law thin-film flow over topography[1] , SYMPHONY CHAKRABORTY, USHA RANGANATHAN, Department of Mathematics, IIT Madras, Chennai 600036, Tamilnadu, India — We consider a thin film of a power-law fluid flowing over an undulated substrate under the action of gravity. Instead of determining the free surface position as in the case of the direct problem, we focus on the inverse problem where for a specific free surface shape, we find the corresponding bottom topography which causes the free surface profile. As an asymptotic approach for thin films and moderate Reynolds numbers, we apply the WRIBL method which enables us to derive a set of two evolution equations for the film thickness h and the flow rate q. We obtain the steady solutions of the above model equation for the inverse problem for weakly undulated free surface profile by a perturbation method. For a strongly undulated free surface shape, we solve the model equation numerically and obtain the bottom topography. We examine the influence of of viscosity of fluid, inertia, film thickness, hydrostatic pressure and surface tension on the reconstructed bottom topography for shear-thinning as well as shear-thickening fluids. The results reveal that compared to shear-thickening fluid, wavy free surfaces for shear-thinning fluid require strongly undulated topographies with steep troughs. Parametric studies show that this effect increases with increasing free surface amplitude.

[1]Research partially supported by IIT Madras.

Monday, November 25, 2013 10:30AM - 12:14PM –

Session H10 Instability: Wakes II - Non-Cylindrical Objects and Wind Tunnels 334 - Timothy C. Lieuwen, Georgia Institute of Technology

10:30AM H10.00001 Turbulent axisymmetric swirling wake: equilibrium similarity solution and experiments with a wind turbine as wake generator , MARTIN WOSNIK, NATHANIEL DUFRESNE, University of New Hampshire — An analytical and experimental investigation of the turbulent axisymmetric swirling wake was carried out. An equilibrium similarity theory was derived that obtained scaling functions from conditions for similarity from the equations of motion, leading to a new scaling function for the decay of the swirling velocity component. Axial and azimuthal (swirl) velocity fields were measured in the wake of a single 3-bladed model wind turbine with rotor diameter of 0.91 m, up to 20 diameters downstream, using X-wire constant temperature hot-wire anemometry. The turbine was positioned in the free stream, near the entrance of the UNH Flow Physics Facility, which has a test section of 6m x 2.7m cross section and 72m length. Measurements were conducted at different rotor loading conditions with blade tip-speed ratios up to 2.8. At U_∞=7 m/s, the Reynolds number based on turbine diameter was approximately 5 $\times 10^5$. Both mean velocity deficit and mean swirl were found to persist beyond 20 diameters downstream. First evidence for a new scaling function for the mean swirl, $W_{max} \propto U_o^{3/2} \propto x^{-1}$ was found. The similarity solution thus predicts that in the axisymmetric swirling wake mean swirl decays faster with x^{-1} than mean velocity deficit with $x^{-2/3}$.

10:43AM H10.00002 Experimental Investigation of Very Large Model Wind Turbine Arrays , KYLE CHARMANSKI, MARTIN WOSNIK, University of New Hampshire — The decrease in energy yield in large wind farms (array losses) and associated revenue losses can be significant. When arrays are sufficiently large they can reach what is known as a fully developed wind turbine array boundary layer, or fully developed wind farm condition. This occurs when the turbulence statistics and the structure of the turbulence, within and above a wind farm, as well as the performance of the turbines remain the same from one row to the next. The study of this condition and how it is affected by parameters such as turbine spacing, power extraction, tip speed ratio, etc. is important for the optimization of large wind farms. An experimental investigation of the fully developed wind farm condition was conducted using a large array of porous disks (upstream) and realistically scaled 3-bladed wind turbines with a diameter of 0.25m. The turbines and porous disks were placed inside a naturally grown turbulent boundary layer in the 6m x 2.5m x 72m test section of the UNH Flow Physics Facility which can achieve test section velocities of up to 14 m/s and Reynolds numbers $\delta^+ = \delta u_\tau/\nu \approx 20,000$. Power, rate of rotation and rotor thrust were measured for select turbines, and hot-wire anemometry was used for flow measurements.

10:56AM H10.00003 Stability of a wind turbine wake subject to root vortex perturbations[1] , DAVID SMITH, HUGH BLACKBURN, JOHN SHERIDAN, Monash University — Results for DNS of a wind turbine wake will be presented. The Tjæborg wind turbine geometry is modelled using a spectral element solver in coupled to an actuator line model described by Sørensen and Shen (2002). The actuator line model considers the flow over the turbine by calculating body forces derived from two-dimensional airfoil data and flow velocity localised at the blade. Using such a model, Ivanell et al. (2010) identified instabilities in the tip vortex for sinusoidal perturbations that reduced the streamwise spacing between tip vortices. In work to be presented we consider perturbations to the blade-root vortex of the turbine. We examine whether perturbations to the root vortex can excite instability mechanisms in the tip vortex and potentially modify tip vortex downstream extents. We also explore how changes to the spacing between root and tip vortices modifies these effects. Ivanell et al. (2010) *J Wind Energy* **13**, Sørensen and Shen. (2002) *J Fluids Eng* **124**.

[1]Supported by Australian Research Council grant DP1096444.

11:09AM H10.00004 Base Flow Asymmetry Effects on the Absolute Stability of Non-uniform Density Wakes , BENJAMIN EMERSON, DAVID NOBLE, TIM LIEUWEN, Georgia Institute of Technology — This work investigates the hydrodynamic stability of bluff body wakes with non-uniform mean density. Such flows are common in bluff body combustors. The absolute/convective stability characteristics of the wake are important, because vortex shedding from the bluff body participates in such processes as mixing, flame blowoff, and combustion instability. Non-uniform density is a sensitive stability parameter for wake flows. Reduction of the wake density relative to the free stream density can stabilize the flow and suppress coherent vortex shedding. Practical bluff body combustors operate at a range of flame density ratios spanning this stability limit. Recent experimental bluff body combustor work by Tuttle et al. investigates wakes with asymmetry in the base flow density profiles. This motivates a hydrodynamic stability model for non-uniform density wakes that includes base flow asymmetry. The model developed in this study investigates the effects of asymmetric base flow velocity and density profiles. It begins with a parameterization of the base flow asymmetries. Results show that base flow asymmetry influences the absolute stability of the flow, and has a strong effect on the most amplified mode shape. The investigation concludes with a comparison to the vorticity equation. Here, we elucidate the physics of the model, and comment on the limitations of such a model.

11:22AM H10.00005 The mechanisms of convective and standing wave mode generation in the wake behind very slender asisymmetric bodies by selective excitation of unstable helical modes , JOSEPH T.C. LIU, KESEOK LEE, School of Engineering, Brown University — Experiments of Asai, et al. (2011) confirm earlier experiments of Sato & Okada (1966), Peterson & Hama (1976) that, for sufficiently slender axisymmetric bodies of revolution placed in a stream parallel to the axes, only convectively unstable modes exist. However, in the downstream nonlinear region, the present theoretical/computational work shows that the imposition of the most unstable helical modes results in the generation of a stationary harmonic-helical mode that persists downstream. This is elucidated from energy transfer mechanism from the mean flow and inter-mode energy transfer via triad interactions. While absolute unstable modes behind bluff bodies of revolution are a natural occurrence according to the linear theory, the presence of such modes behind very slender bodies of revolution is a consequence of downstream nonlinear interactions between the excited helical modes.

11:35AM H10.00006 ABSTRACT WITHDRAWN –

11:48AM H10.00007 Experimental study of the interaction between the horseshoe system and the vortex shedding of a wall-mounted rectangular cylinder , MOUHAMMAD EL HASSAN, ROBERT MARTINUZZI, University of Calgary — The interaction between the horseshoe vortex system (HVS) and the shedding of large-scale vortical structures of a wall-mounted rectangular cylinder (height-to-width ratio h/d = 4) is investigated experimentally for both a canonical and a perturbed boundary layers (δ/h = 0.18 and 0.64, respectively). The Reynolds number is $Re_d = 12000$. The 3D flow is reconstructed from uncorrelated 2D snapshots of time-resolved Particle Image Velocimetry data, using proper orthogonal decomposition, a phase-averaging technique and symmetry/antisymmetry decomposition of the flow. It is found that the dynamics of the HVS affects the topology of the vortex shedding near the wall, particularly for the thicker boundary layer. The back-flow and the zero-flow modes of the HVS have particular influence on the symmetry of the horseshoe legs and its momentum content. The orientation and the momentum content of the horseshoe legs can result in the bend of the shedding vortices towards the cylinder back wall. An interaction between the tip and the junction flows is also observed just downstream from the obstacle. A downwash mechanism and a like eruption process are evidenced for the natural boundary layer whether a strong upwash dominate the tip-junction interaction for the thicker boundary layer.

12:01PM H10.00008 Path instabilities of heavy bodies in free fall in a viscous fluid: wake dynamics vs. aerodynamic effects , DAVID FABRE, KAMAL SELVAM, JOËL TCHOUFAG, IMFT, University of Toulouse, PAULINE ASSEMAT, Monach, Australia, JACQUES MAGNAUDET, IMFT, University of Toulouse — Solid bodies in free fall in a viscous fluid generally fall along a non-straight path, and a variety of periodic (fluttering, tumbling) and non-periodic regimes can be observed. We analyze the structure of the couplings between the fluid and the body, restricting to a linear stability framework. Introducing a simple toy model consisting of a infinitely long plate sliding along a vertical wall, we show that in the limit of large solid-to-fluid masses a decoupling takes place, allowing us to distinguish two kinds of modes: "wake" modes in which the body motion has virtually no influence, and "body" modes for which the intrinsic wake dynamics can be neglected. Turning to more realistic objects, we show that the "body" modes can be described through a rationally derived aerodynamic model (based on quasi-static assumptions), yielding either a static instability, or a dynamic, low-frequency, instability. Considering 2D rectangular rods and 3D disks, we explore the competition between the three kinds of instabilities. For objects elongated in the spanwise direction, it is found that wake instability dominates in case of 2D rectangles and low-frequency instability dominates in case of disks. For objects elongated in the streamwise direction, static instability always dominate.

Monday, November 25, 2013 10:30AM - 12:27PM –

Session H11 Bubbles V: Rising Bubbles and Surface Interaction 335 - Omar K. Matar, Imperial College of London

10:30AM H11.00001 Interaction of two oscillating bubbles rising in a thin gap , AUDREY FILELLA, VERONIQUE ROIG, PATRICIA ERN, IMFT — We investigate experimentally different mechanisms of hydrodynamic interaction between two oscillating bubbles rising in a liquid at rest confined in a thin gap cell. In order to understand the relation between the motion of a single bubble and its wake as well as the hydrodynamic interaction between two oscillating bubbles, we characterize the kinematics of the bubbles interaction using high speed cameras and we measure the associated liquid velocity by High-Frequency PIV. The motion of the trailing bubble is modified while passing through the unstable wake of the leading bubble: depending on the relative size of the bubbles and their relative position, we can observe horizontal attraction to the center of the leading bubble's wake, vertical entrainment, ejection of the second bubble by a vortex of the wake or a sequential combination of the three mechanisms.

10:43AM H11.00002 The effect of surfactants on path instability of a rising bubble[1] , YOSHIYUKI TAGAWA, Tokyo University of Agriculture and Technology, SHU TAKAGI, YOICHIRO MATSUMOTO, The University of Tokyo — We experimentally investigate the surfactant effect on path instability of an air bubble rising in quiescent water. An addition of surfactant varies the gas-water boundary condition from zero shear stress to non-zero shear stress. We report three main findings: firstly, while the drag force acting on the bubble increases with the surfactant concentration as expected, the lift force shows a non-monotonic behavior; secondly, the transient trajectory starting from helical to zigzag is observed, which has never been reported in the case of purified water; lastly, a bubble with the intermediate slip conditions between free-slip and no-slip show a helical motion for a broad range of the Reynolds number. Aforementioned results are rationalized by considering the adsorption-desorption kinetics of the surfactants on gas-water interface and the wake dynamics.

[1]Y.T. thanks for financial support from Grant-in-Aid for JSPS Fellows (20-10701). We also thank for Grant-in-Aid for Scientific Research (B) (21360079).

10:56AM H11.00003 Bubble rise in a non-isothermal channel with a non-monotonic dependence of the surface tension on temperature , KHELLIL SEFIANE, University of Edinburgh, MANOJ TRIPATHI, KIRTI SAHU, Indian Institute of Technology, Hyderabad, GEORGE KARAPETSAS, University of Thessaly, OMAR MATAR, Imperial College London — We examine the dynamics of a bubble rising inside a non-uniformly heated, vertically-aligned channel. A constant temperature gradient is imposed at the channel walls. We use a diffuse-interface formulation, and solve the continuity, Navier-Stokes and energy equations coupled with Cahn-Hilliard equations. A non-monotonic dependence of the surface tension on temperature is adopted, which exhibits a well-defined minimum. We focus on the effect of this functional dependence on the bubble rise dynamics, and investigate the influence of the minimum depth on the velocity and temperature fields, bubble motion and deformation. We present the results of a full parametric study of the flow dynamics, and compare our numerical predictions to experimental data.

11:09AM H11.00004 Numerical simulation of a bubble rising in an unconfined viscoplastic fluid with chemical reaction , MANOJ TRIPATHI, KIRTI SAHU, Indian Institute of Technology, Hyderabad, GEORGE KARAPETSAS, University of Thessaly, OMAR MATAR, Imperial College London — We investigate the flow dynamics of a rising bubble under the action of gravity surrounded by an unconfined Herschel-Bulkley fluid; a second-order chemical reaction is also taken into account. We solve the continuity, Navier-Stokes and energy equations coupled with Cahn-Hilliard equations associated with the mole fraction of the reactants and product. Interfacial forces are accounted for in the Navier-Stokes equations, and a Papathanasiou model is used to incorporate yield stress effects into the governing equations. The effects of various dimensionless groups, such as the Reynolds, Bond, Bingham, and Damkohler numbers, on the flow are investigated in terms of flow structure, concentration, velocity and temperature fields, bubble deformation, and unyielded regions. Our results for the unreactive case have been validated against previous work. The work on the reactive case is under progress and a parametric study of this investigation will be discussed at the meeting.

11:22AM H11.00005 Effects of surfactant and electrolyte on the drainage of the thin liquid film between a glass plate and a bubble approaching at a constant velocity , SAORI SHIMOYAMA, TOSHIYUKI OGASAWARA, HIROYUKI TAKAHIRA, Dept. of Mechanical Engineering, Osaka Prefecture University, FLUID ENGINEERING LABORATORY TEAM — The drainage process of the thin liquid film between a glass plate and a bubble in the presence of impurities in water has been investigated experimentally. An electrolyte solution and a surfactant solution are produced by the addition of $MgSO_4$ and Triton X-100 into super purified water, respectively. The hemispherical bubble is generated at a tip of a pipe and translates toward the flat glass surface at constant approaching velocities ranging from 1 μm/s to 5000 μm/s. The thickness distribution of the liquid film formed between two surfaces is measured by the laser interferometer and fringe patterns are recorded by a high-speed video camera. In 0.05 M Triton X-100 solution, the bubble surface becomes no-slip condition due to the Marangoni effect, which delays the film drainage. Hence, deeper dimple shape is formed even at lower approaching velocities less than 1000 μm/s. On the other hands, in super purified water and 0.5 M $MgSO_4$ solution, the film does not form clear dimple shape and ruptures in the early stage of the interaction at lower approaching velocities. However, as the approaching velocity increases, the dimple shape becomes deeper and wider in $MgSO_4$ solution than that in super purified water although the bubble surfaces are free-slip condition in both cases.

11:35AM H11.00006 Experimental study of the interaction of a bubble with an inclined wall , C. BARBOSA, R. ZENIT, Universidad Nacional Autonoma de Mexico, D. LEGENDRE, Institut de Mecanique des Fluides de Toulouse — Bubbly flows are very common in many engineering applications and natural flows. The interaction of bubbles with containing walls is very important to understand the properties of the bulk flow. In this study we analyse the interaction of a single air bubble with an inclined wall. Experiments are conducted in a rectangular container in which bubbles are generated using capillary tubes of different diameters. The bubbles collide with a glass wall that can be adjusted to vary its inclination. The motion of the bubble is filmed with a high speed video camera. We observe that, for a horizontal wall, the bubbles collide and bounce repeatedly before arresting against the surface, in accordance with previous studies. For inclined walls, after the collision-rebound process, the bubbles slide over the wall reaching a terminal velocity which depends on the angle of inclination, bubble size and liquid properties. We discuss the nature of the bouncing process and the wall-induced drag for a wide variety of conditions.

11:48AM H11.00007 On the thickness of a film generated during a topological rearrangement , PAULINE PETIT, ILM, UMR5306, Université Lyon 1, France, JACOPO SEIWERT, ISABELLE CANTAT, IPR, UMR6251, Université de Rennes 1, France, ANNE-LAURE BIANCE, ILM, UMR5306, Université Lyon 1, France — T1 topological rearrangement, i.e. switching of neighboring bubbles in a liquid foam, is the elementary process of foam dynamics. It has been extensively studied as it is a crucial point for foam rheology [Cohen-Addad *et al.*, 2013] or foam collapse [Biance *et al.*, 2011]. The dynamic of T1 has been proved to depend a lot on the surfactants used in the foaming process, and different modeling taking into account surface viscosity and/or elasticity have been proposed [Durand *et al.*, 2006; Biance *et al.*, 2009; Grassia *et al.*, 2012]. By performing experiments in a cubic assembly of films, we go a step forward this global analysis in investigating the structure of the freshly formed film. In particular, the flow velocity field is probed by particle tracking and the film thickness by light absorption and interferometric measurements. Two distinct behaviors have been observed: for mobile surfactants, the observations suggest an elongation flow and a stretched convected behavior of the liquid in the film, as for more rigid ones, the liquid in the film is at rest, its structure being entirely governed by its formation near the liquid junction (i.e. Plateau border) connecting the neighboring films.

12:01PM H11.00008 Formation of Gas Pockets in a Boundary Layer Under Turbulent Forcing[1] , FRANCISCO PEREIRA[2], DAVID JEON, MORTEZA GHARIB, California Institute of Technology — Our experiments examine the formation of gas pockets in a flat plate boundary layer in water submitted to turbulent forcing. Air micro-bubbles are injected from wall orifices over surfaces with hydrophilic, hydrophobic and super-hydrophobic wetting properties. In this latter case, the surfaces are coated with paints with water repellent characteristics, or covered with vertically aligned carbon nanotubes (CNTs) that are chemically tuned to produce super-hydrophobicity. The turbulence level of the incoming flow is adjusted through meshes and grids. Bubble injection is operated at constant flow rate, and the mechanism of bubble growth and detachment is investigated through high speed recordings. Super-hydrophobicity is found to promote the attachment of discrete gas voids to the wall, with CNTs favoring the formation of a gas layer. Another finding is that the turbulence intensity affects the stability of microbubbles attached to the wall under super-hydrophobic conditions.

[1]This work is supported by ONR Grant N00014-11-1-0031

[2]FP was on unpaid leave from CNR-INSEAN (Rome, Italy)

12:14PM H11.00009 Void fraction and bubble size in a simulated hydraulic jump[1] , ADAM WITT, JOHN GULLIVER, LIAN SHEN, St. Anthony Falls Laboratory — Two- and three-dimensional numerical simulations of a hydraulic jump are carried out with the open source software package OpenFOAM using a Volume of Fluid numerical method and a realizable k-ε turbulence model. Time-averaged air-water properties are obtained over a 15 second sampling time. Void fraction profiles show good agreement with experimental values in the turbulent shear layer. Sauter mean diameter approaches experimental results in the turbulent shear layer, while showing grid dependence down to a uniform computational cell size of 0.625 mm. Three-dimensional results show a minor improvement in the prediction of entrained air compared to two-dimensional results at a multiple of 341 in increased computational time for the chosen grid. Relative error in bubble diameter is similar between two- and three-dimensional simulations. The results indicate a Volume of Fluid, realizable k-ε numerical model accurately predicts the void fraction profile when the Sauter mean diameter to grid size ratio surpasses 8.

[1]This research was supported by funding from the U.S. Department of Energy, the Hydro Research Foundation, the University of Minnesota and the University of Minnesota Supercomputing Institute.

Monday, November 25, 2013 10:30AM - 12:27PM –

Session H12 Vortex Dynamics and Vortex Flows VI 336 - William Irvine, University of Chicago

10:30AM H12.00001 Dynamics of Quasi 2D and 3D Co-rotating Vortex Merger , AKSHAY KHANDEKAR, JAMEY JACOB, Oklahoma State University — Merger of vortices is examined experimentally to compare the merger of slender parallel vortices generated either coincidentally or continuously. It is known that like-sign vortices rotate around a common center of circulation and merger between the vortices may occur under certain conditions. This merger is dependent on the strength of the vortex circulation, distance of separation between the centers of the two vortices, Re_Γ, and vorticity distribution. Quasi-2D and 3D experimental data is examined and merger relations are derived. The former uses high aspect ratio rotating paddles in a tank and while the latter are from wing-tip vortices in a wind tunnel. The vortex merger tank generates slender co-rotating vortices and are examined using PIV, while in the wind tunnel two opposing wings are arranged at opposite angles of attack to generate a pair of vortices that merge downstream. A 5-hole probe is used to obtain 3D velocity vectors via wake survey, along with PIV. The procedure is performed in the wake at different distances to observe merger under different conditions. Temporally and spatially dependent relations in quasi-2D and 3D vortex merger are derived. Merger behavior is generally similar between the cases, but instabilities along quasi-2D vortices may affect

10:43AM H12.00002 Interaction of a vortex ring with a natural convective layer , C.A. PALACIOS-MORALES, M. SALINAS, F.J. SOLORIO-ORDAZ, R. ZENIT, Universidad Nacional Autonoma de Mexico — We study the dynamics and heat transfer resulting from the impact of a vortex ring with a vertical heated wall. Laminar vortex rings were generated in water with a piston- cylinder arrangement. The vertical wall is heated by a thermal bath which is held at constant temperature producing a laminar and stable thermal boundary layer. Measurements of the 2D velocity field were obtained with a TR-PIV technique and the scalar temperature field is obtained by the PLIF technique. To avoid azimuthal instabilities, we conducted experiments for small stroke rations and Re of O(1000). The initial circular shape evolves to an asymmetric shape after reaching the wall. The lower ring section thickens and separates from the wall while the upper part thins and is dragged by the thermal layer. On the sides, the vortex ring is stretched. The rate of change of circulation is small at the lower section of the ring indicating that the momentum transport and heat transfer is more significant in this region. The instantaneous heat transfer coefficient was obtained; as expected, when the vortex approaches the wall, the heat transfer increases mainly at the lower part of the ring.

10:56AM H12.00003 Interactions of two unequal co-rotating viscous vortices in the presence of external shear , PATRICK FOLZ, KEIKO NOMURA, University of California, San Diego — The interaction of two co-rotating viscous vortices in linear background shear is investigated through two-dimensional numerical simulations. In general, equal co-rotating viscous vortices will merge if brought within a critical separation distance. This process occurs when the relative strain induced on one vortex by the other weakens it sufficiently, causing core fluid to detrain. The vortices are then mutually entrained and the flow transforms into a single vortex. In the absence of shear, when the vortices are unequal the outcome of the interaction is determined by the relative timing of core detrainment. Depending on the degree of asymmetry, merger may or may not occur. When background shear is present, advective motion of the vortices is altered. With sufficiently strong adverse shear, the vortices will separate. Otherwise, in the case of equal vortices, merger is enhanced or inhibited by favorable or adverse shear respectively. The onset of the merging process when shear is present is found to occur when the vortices reach the critical merging criterion for vortices without shear. For unequal vortices, the presence of the shear modifies the start of the detrainment process for each vortex and leads to varied outcomes.

11:09AM H12.00004 Investigation of the Unsteady Total Pressure Profile Corresponding to Counter-Rotating Vortices in an Internal Flow Application , KATHRYN GORDON, SCOTT MORRIS, ALEKSANDAR JEMCOV, JOSHUA CAMERON, University of Notre Dame — The interaction of components in a compressible, internal flow often results in unsteady interactions between the wakes and moving blades. A prime example in which this flow feature is of interest is the interaction between the downstream rotor blades in a transonic axial compressor with the wake vortices shed from the upstream inlet guide vane (IGV). Previous work shows that a double row of counter-rotating vortices convects downstream into the rotor passage as a result of the rotor blade bow shock impinging on the IGV. The rotor-relative time-mean total pressure distribution has a region of high total pressure corresponding to the pathline of the vortices. The present work focuses on the relationship between the magnitude of the time-mean rotor-relative total pressure profile and the axial spacing between the IGV and the rotor. A survey of different axial gap sizes is performed in a two-dimensional computational study to obtain the sensitivity of the pressure profile amplitude to IGV-rotor axial spacing.

11:22AM H12.00005 Analysis of the formation and evolution of vortex rings in non Newtonian fluids using 3D PTV[1] , ABHISHEK BAJPAYEE, ALEXANDRA TECHET, Massachusetts Institute of Technology — Formation and evolution of vortex rings have been studied for a long time but mostly only in Newtonian fluids. However, many fluids in nature and in the industry such as blood, crude oil, etc., exhibit non Newtonian characteristics. Palacios-Morales and Zenit recently studied the formation of vortex rings in shear thinning liquids for the first time using 2D PIV and compared experimental findings with theoretical predictions. The authors recently demonstrated the applicability of Light Field (LF) imaging to conduct 3D Particle Tracking Velocimetry (PTV) to study densely seeded flow fields and their evolution over time using synthetic data. LF based 3D PTV is now used to quantitatively study vortex rings created in Glycerin based on multiple parameters and the results are compared with previous findings.

[1]ONR (Grant #N00014-12-1-0787, Dr. Steven Russell), Naval Engineering Education Center

11:35AM H12.00006 Buoyancy-Induced Columnar Vortices[1] , MARK SIMPSON, ARI GLEZER, Georgia Institute of Technology — The formation of anchored, buoyancy-driven columnar vortices that is driven by the instability of a thermally stratified air layer and is sustained by entrainment of ground-heated air is investigated in a meter-scale laboratory facility using a heated ground plane and an azimuthal array of flow vanes. Naturally-occurring, buoyancy-driven columnar vortices ("dust devils") spontaneously occur with core diameter of 1-50 m at the surface and heights up to one km, with considerable angular and axial momentum. Such vortices convert low-grade waste heat in an air layer overlying a warm surface into a flow with significant kinetic energy. The considerable kinetic energy of the vortex column cannot be explained by buoyancy alone and is a result of the production, concentration, and tilting of horizontal vorticity produced in the air layer over the heated ground plane. The present investigation focuses on the fundamental mechanisms of the formation, evolution, and dynamics of the available vorticity within the columnar vortex using stereo-PIV with specific emphasis on the scaling and distribution of the available kinetic energy flux. It is shown that the scaling and strength of these vortices can be significantly altered through adjustments of the flow vanes and the global sensible heat absorbed by the air flow.

[1]Supported by ARPA-E

11:48AM H12.00007 ABSTRACT WITHDRAWN –

12:01PM H12.00008 Numerical simulation of a spanwise vortex in a tidal induced flow[1] , ERICK J. LOPEZ-SANCHEZ, GERARDO RUIZ-CHAVARRIA, Facultad de Ciencias, Universidad Nacional Autonoma de Mexico — The system formed by two counter-rotating vortices (known as a dipole) occurs often in geophysical flows and it has been the subject of some experimental and numerical investigations. In some previous works (e. g. Lacaze et. al. *Exp. Fluids* **48** (2010) 225-231) a spanwise vortex in front of the dipole has been observed. In this work we study the evolution of this transversal vortex in a system consisting of a channel flushing into a open domain and subject to a periodic forcing. To this end the Navier-Stokes and continuity equations are solved with a finite volume code (OpenFOAM). The numerical solution has been obtained for a Reynolds number $Re = 1000$ and two different values of the Strouhal number, namely $S = 0.01$ and $S = 0.02$, for which the dipole moves away from the channel. In addition two different aspect ratios (depth to channel width) are considered, these are 0.5 and 1. The horseshoe vortex forms near the bottom and it lifts as the dipole moves. We compare the evolution of the spanwise vortex with respect the case of a vortex produced by a impulsively jet and we highlight the effects of the periodic forcing. Finally we compare our results with some experimental data obtained in laboratory.

[1]Authors acknowledge support by DGAPA-UNAM under project IN116312 "Vorticidad y ondas no lineales en fluidos"

12:14PM H12.00009 On Clarifying the Mechanisms for Persistent Asymmetries in Advecting Vortical Motions , JOHN ELSNAB, HURMAT UI AIN, University of Melbourne, JOSEPH KLEWICKI, University of Melbourne and University of New Hampshire — A challenge associated with the study of turbulence relates to determining how and why ensembles of instantaneous motions underlie the observed behaviors of the time averaged flow. It is important to distinguish between events that make a lasting and unique signature to the time average representation of the flow, and events that simply make instantaneous contributions. The present experiments establish when laminar vortex rings interact with a time evolving shear-layer that persistent asymmetries are generated. These asymmetries are dynamically significant as they modify the gradient of the Reynolds stress (RS), which is the relevant quantity that appears in the mean equations. In turbulent wall flows, the gradient of the RS acts as a net source or sink of mean momentum depending upon the position where the RS is maximum. Vortex rings subjected to an induced advection velocity (an additive perturbation) do not exhibit persistent asymmetries; however, when rings with modified advection velocities interact with a shear-layer, the ring dynamics exhibit enhanced asymmetries when compared to shear-layer interactions alone. Connections are drawn between the velocity and vorticity field correlations that are attributed to the RS gradient and its modification.

Monday, November 25, 2013 10:30AM - 12:27PM –

Session H13 Granular Flows II: Applications 301 - Nicholas Pohlman, Northern Illinois University

10:30AM H13.00001 Erosion and flow of hydrophobic granular materials[1] , BRIAN UTTER, THOMAS BENNS, JOSEPH MAHLER, James Madison University — We experimentally investigate submerged granular flows of hydrophobic and hydrophilic grains both in a rotating drum geometry and under erosion by a surface water flow. While slurry and suspension flows are common in nature and industry, effects of surface chemistry on flow behavior have received relatively little attention. In the rotating drum , we use varying 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. High concentrations of hydrophobic grains result in an effectively cohesive interaction between the grains forming aggregates, with aggregate size and repose angle increasing with hydrophobic concentration. However, the formation and nature of the aggregates depends significantly on the presence of air in the system. We present results from a related experiment on erosion by a surface water flow designed to characterize the effects of heterogeneous granular surfaces on channelization and erosion.

[1]Supported by NSF CBET Award 1067598.

10:43AM H13.00002 Granular dynamics of the low fill regime in a cylindrical tumbler: particle-wall slip[1] , DENNIS IVAN W. DIAZ, PAUL B. UMBANHOWAR, JULIO M. OTTINO, RICHARD M. LUEPTOW, Northwestern University — Investigations of granular flow in tumblers generally avoid low fill fractions due to the absence of a well-defined flowing layer and the high amount of slip between particles and the tumbler wall. Motivated by results from spherical and double-cone tumblers that exhibit slow axial drift and shallow layers near the "poles," we explored the low fill fraction regime using video analysis to track the particle motion. Large angular amplitude slip occurred at low fill levels and rotation rates. As fill level was increased, the mean time between slip events decreased as did the slip amplitude until, above a critical fill level, particle-wall slip effectively ceased. The mean time between slip events at fixed fill level was reduced by both increasing the tumbler rotation rate and decreasing the particle diameter. We use our results to evaluate the relationship between particle-wall slip and formation of a flowing layer.

[1]Funded by NSF Grant CMMI-1000469

10:56AM H13.00003 The effects of soft-sphere contact models on heat transfer to particles flowing over a heated surface , AARON MORRIS, CHRISTINE HRENYA, The University of Colorado at Boulder, ZHIWEN MA, National Renewable Energy Laboratory, SREEKANTH PANNALA, Oakridge National Laboratory, TOM O'BRIEN, Retired — DEM simulations are performed for solid particles flowing around a heated surface. For moderately dense granular flows with enduring particle-wall contacts, particles in contact with the surface are warmed by conduction across the mutual contact area. Heat transfer may also occur via conduction through the interstitial fluid within the small gaps between particles and the wall. The conductive heat transfer depends on the specific contact model, i.e. Hertzian or linear spring dashpot (LSD), because such models determine the contact area and duration. In this work, we use MFIX DEM (an open source simulation tool developed at NETL) to simulate particles falling in crossflow around a heated cylinder. Heat transfer models for both contact conduction as well as conduction across the interstitial fluid are included in these simulations. We discuss how different collision models impact the heat transfer to the particles as well as the sensitivity to various model parameters. We also compare the heat transfer predicted by different contact conduction thermal models.

11:09AM H13.00004 Stagnation, circulation, and erosion of granular materials through belt conveyor sluice gate[1] , NICHOLAS POHLMAN, MICHAEL MORALDA, Northern Illinois University, RYAN DUNNE, Humboldt State University — Control of flow rates in conversion reactors for discrete materials like biomass can be achieved in belt conveyors through a combination of belt speed, hopper size, and aperture opening. As material is extracted from the bottom of the storage hopper, other material cannot achieve plug flow and therefore is restricted from exiting through a sluice-gate type opening. The excess material moves vertically from the opening causing a pile up and recirculation back along the free surface of the hopper. Experimental results obtained through high speed imaging show the position of the stagnation point as well as the rate of circulation is dependent on the mass flow rate achieved and instantaneous fill level. The movement of material into the plug flow along the belt allows verification of deposition models on erodible beds rather than rigid surfaces with artificial roughness of glued particles. Similarly, the pile-up at the exit influences the efficiency of the transport affecting the narrow energy return on investment of biomass resources. The laboratory-scale behavior can therefore be translated into industrial performance metrics for increased operational efficiency.

[1]This work is supported by the NSF REU Site *Operation E-Tank* under award number 1156789.

11:22AM H13.00005 Incorporation of Interstitial Gas Effects on Granular Flows , CHRISTINE HRENYA, University of Colorado, VICENTE GARZO, Universidad de Extremadura, SUDHEER TENNETI, SHANKAR SUBRAMANIAM, Iowa State University — Numerous examples of granular flows exist in which the role of the interstitial gas cannot be ignored. A range of approaches have been taken to incorporate these effects into continuum descriptions. Early efforts simply added a mean drag law to the momentum balance. This ad hoc approach was followed by more rigorous treatments in which an instantaneous drag was incorporated directly into the kinetic equation. Analytical expressions for the resulting continuum description were obtained in the Stokes limit, but not possible higher Reynolds numbers. In the current effort, DNS-based simulations are used to develop a model for the instantaneous drag force that is applicable to a wide range of Reynolds number. This model, based on the Langevin equation, is incorporated into the Enskog equation in order to derive a continuum description for the gas-solid flow. In the limit of Stokes flow, the additional terms arising in the conservation equation are found to match those of previous analytical treatments. Furthermore, the impact of gas on the solid-phase constitutive relations, which was ignored in analytical treatments, is determined. The parameter space examined is consistent with that found in circulating fluidized beds. For such systems, the results indicate a non-negligible impact of the gas phase on the shear viscosity and the Dufour coefficient.

11:35AM H13.00006 Experimental and computational investigation on the flow behavior of granular particles through an inclined rotating chute , SUSHIL SHIRSATH, JOHAN PADDING, HERMAN CLERCX, HANS KUIPERS, Eindhoven University of Technology, Eindhoven, Netherlands — In blast furnaces operated in the steel industry, particles like coke, sinter and pellets enter from a hopper and are distributed on the burden surface by a rotating chute. Such particulate flows suffer occasionally from particle segregation in chute, which hinders efficient throughflow. To obtain a more fundamental insight into these effects, monodisperse particles flowing through a rotating chute inclined at a fixed angle has been studied both with experiments and with a discrete particle model. We observe that the prevailing flow patterns depend strongly on the rotation rate of the chute.With increasing rotation rate the particles are moving increasingly to the side wall.The streamwise particle velocity is slightly reduced in the first half length of the chute due to the Coriolis force, but strongly increased in the second half due to the centrifugal forces.The particle bed height becomes a two-dimensional function of the position inside the chute, with a strong increase in bed height along the sidewall due to the Coriolis forces. It was found that the DPM model was agreed well with the experimental measurements.We will also discuss ongoing work, where we investigate the effects of binary particle mixtures with different particle size or density, different chute geometry.

11:48AM H13.00007 Collapsing granular beds: The role of interstitial air , DEVARAJ VAN DER MEER, CHRISTA GJALTEMA, TESS HOMAN, Physics of Fluids group, University of Twente, the Netherlands — A pre-fluidized sand bed consisting of fine particles compactifies when it is subjected to a shock. We find that the response depends on both the shock strength and the ambient pressure where, counterintuitively, the bed height decreases less at lower ambient pressures. We investigate what happens to the interstitial air during compaction by measuring the pressure variations above and below the bed: The top pressure is observed to decrease abruptly –on the time scale of the compaction– whereas that below the bed very slowly rises to a maximum. Subsequently both pressures slowly relax to ambient values. We introduce a one-dimensional diffusion model that uses only the change in bed height and the ambient pressure as an input, and find that it accurately accounts for the measured pressure variations.

12:01PM H13.00008 Avalanche to Continuous flow transition in wet and cohesive granular media[1] , ASHISH ORPE, SAPRATIV BASU, PANKAJ DOSHI, Chemical Engineering Division, National Chemical Laboratory — We have studied the flow of wet and cohesive granular media in a partially filled, horizontally rotating cylinder. Very small, amount of viscous liquid is added to dry granular particles and the mixture is rotated in the cylinder at various rotational speeds to determine the angle of repose in the avalanching regime, the continuous regime and at the transition rotational speed separating the two regimes. Every experimental run is carried out afresh at a pre-defined rotational speed using liquids with different free surface tension and added in different amounts. Increasing the liquid surface tension increases the angle of repose as well as shifts the transition rotational speed to increasingly higher values. Similar qualitative behaviour is also observed on increasing the amount liquid added. A linear dependence is observed when the transition angle of repose for all cases is plotted against the corresponding transition rotational speed. The entire flow regime is modeled using momentum and mass balance equations for the flowing layer of particles. The total stress in the flowing mass of particles is assumed to be a linear combination of frictional, collisional and capillary force contributions. The model equations are able to reproduce most of the observed flow behavior.

[1]Department of Science and Technology, India, (Grant No. SR/S3/CE/037/2009)

12:14PM H13.00009 ABSTRACT WITHDRAWN –

Monday, November 25, 2013 10:30AM - 12:40PM –

Session H14 Experimental Techniques V: Two-Phase/Ablation 302 - Douglas G. Bohl, Clarkson University

10:30AM H14.00001 Comparison of Global Sizing Velocimetry and Phase Doppler Anemometry measurements of alternative jet fuel sprays[1] , REZA SADR, KUMARAN KANNAIYAN, Texas A&M University at Qatar — Atomization plays a crucial precursor role in liquid fuel combustion that directly affects the evaporation, mixing, and emission levels. Laser diagnostic techniques are often used to study the spray characteristics of liquid fuels. The objective of this work is to compare the spray measurements of Gas-to Liquid (GTL) jet fuels obtained using Global Sizing Velocimetry (GSV) and Phase Doppler Anemometry (PDA) techniques at global and local levels, respectively. The chemical and physical properties of GTL fuels are different from conventional jet fuels, owing to the difference in their production methodology. In this work, the experimental facility, the measurement techniques, and spray characteristics of two different GTL fuels are discussed and compared with those of Jet A-1 fuel. Results clearly demonstrate that although the global measurement gives an overall picture of the spray, fine details are obtained only through local measurements and complement in gaining more inferences into the spray characteristics. The results also show a close similarity in spray characteristics between GTL and Jet A-1 fuels.

[1]Funded by Qatar Science and Technology Park.

10:43AM H14.00002 Dark Field Imaging of Multiphase Fluid Flows , BARRY SCHARFMAN, ALEXANDRA TECHET, MIT — A novel method has been developed for spatially and temporally resolving three-dimensional multiphase fluid flows. Image volumes are captured using a multiple CCD sensor array consisting of a planar array of cameras. This is similar to light field imaging, but no light enters the cameras directly; rather, lights surround the scene and a dark sheet is placed directly across from the array. Therefore, each camera records an essentially binary image of the scene from a different angle, which simplifies subsequent image processing. Synthetic aperture refocusing techniques are applied to the raw camera array images, each with large depths of field, to obtain a stack of post-processed images, with narrow depth of field, where each image in the stack is located on a specific focal plane. Then, flow features are extracted from the binary refocused volume, allowing the scene to be reconstructed in three dimensions over time. Simulations and experimental fluid flows are used to validate and improve this technique.

10:56AM H14.00003 Naphthalene Planar Laser-Induced Fluorescence Imaging of Orion Multi-Purpose Crew Vehicle Heat Shield Ablation Products[1] , CHRISTOPHER S. COMBS, NOEL T. CLEMENS, The University of Texas at Austin, PAUL M. DANEHY, NASA Langley Research Center — The Orion Multi-Purpose Crew Vehicle (MPCV) calls for an ablative heat shield. In order to better design this heat shield and others that will undergo planetary entry, an improved understanding of the ablation process is required. Given that ablation is a multi-physics process involving heat and mass transfer, codes aiming to predict heat shield ablation are in need of experimental data pertaining to the turbulent transport of ablation products for validation. At The University of Texas at Austin, a technique is being developed that uses planar laser-induced fluorescence (PLIF) of a low-temperature sublimating ablator (naphthalene) to visualize the transport of ablation products in a supersonic flow. Since ablation at reentry temperatures can be difficult to recreate in a laboratory setting it is desirable to create a limited physics problem and simulate the ablation process at relatively low temperature conditions using naphthalene. A scaled Orion MPCV model with a solid naphthalene heat shield has been tested in a Mach 5 wind tunnel at various angles of attack in the current work. PLIF images have shown high concentrations of scalar in the capsule wake region, intermittent turbulent structures on the heat shield surface, and interesting details of the capsule shear layer structure.

[1]This work was supported by a NASA Office of the Chief Technologist's Space Technology Research Fellowship (NNX11AN55H)

11:09AM H14.00004 Optimization of Chemical Concentrations for Molecular Tagging Velocimetry[1] , WYATT SPELLMAN, DOUGLAS BOHL, Clarkson University — Molecular Tagging Velocimetry (MTV) is a whole field optical diagnostic technique where long lived chemical tracers are mixed on a molecular level with the working fluid. The chemical tracer is excited into phosphorescence using a light source, typically a pulsed UV laser. Because MTV is an absorption/emission technique, versus PIV which is a scattering technique, the light levels can be an order of magnitude lower than that for PIV. It is therefore important to optimize the individual chemical concentrations to maximize the emission in the desired field of view. Three chemicals are used in combination to create the molecular tracer in water. As with all absorption techniques, the intensity of the emitted light is a function of the depth of penetration (i.e. the distance the light beam travels through the fluid medium) due to attenuation of the beam. Attenuation is a function of the tracer concentration, which for MTV is a complicated due to the use of three chemical to create the tracer chemical. In this work we investigate the effect of chemical concentration on the attenuation in order to optimize the mixture so that the emission levels are maximized at any specified depth in the fluid.

[1]This work supported by NSF Grant # 0845882.

11:22AM H14.00005 Adsorption of diatoms at the oil-water interface , NILOOFAR FATHOLLAHI, JIAN SHENG, Texas Tech Univ. — Statistically robust experimental observations on 3D trajectory of diatoms approaching an oil-water interface is crucial for understanding sorption mechanisms of active particles, and interfacial rheology with over-arching implications in interfacial dynamics, droplet break and coalescence. Digital Holographic Cinematography is utilized to measure 3-D trajectories of diatoms, *Thalassiosira pseudomona* and *T. weissflogii* and simultaneously track the interface. Experiments are conducted in a 300x100x100mm chamber containing 32 ppt artificial seawater. A stationary pendant drop is created on the tip of a needle located at the center of the chamber. Three oil samples, Louisiana crude, hexadecane, and mineral oil, are used. Diatoms are injected at a height above the drop with a negligible velocity, where Diatom precipitates freely on its excess weight. Holograms of diatom and drop are recorded at 5 fps with a magnification of 1.3X and are streamed in real time allowing for long-term study of sorption onto a slowly aging interface. A novel autofocus algorithm enables us to determine 3D locations within an uncertainty of 0.05 particle diameter. This allows us to perform super-resolution measurement to determine the effects of location and orientation of diatoms on the adsorption rate at the oil-water interface. Funded by GoMRI.

11:35AM H14.00006 Dynamics of particle—turbulence interaction at the dissipative scales[1] , HUMBERTO BOCANEGRA EVANS, Faculty of Applied Physics, Eindhoven Universiy of Technology, NICO DAM, Faculty of Mechanical Engineering, Eindhoven Universiy of Technology, WILLEM VAN DE WATER, Faculty of Applied Physics, Eindhoven University of Technology, JM BURGERSCENTRUM COLLABORATION, COST ACTION, PARTICLES IN TURBULENCE COLLABORATION — We present results of a novel phosphorescent tagging technique that is particularly suited to study particle-laden flows. Using phosphorescent droplets we probe the dynamics of particle–turbulence interaction at the dissipative length scales. We create a cloud of droplets within a chamber capable of generating homogeneous, isotropic turbulence with zero-mean flow. The droplets have Stokes number $\mathrm{St} \sim 1$, and the flow is intensely turbulent, with Reynolds number $\mathrm{Re}_\lambda \approx 500$. Using a frequency-tripled Nd:YAG laser, we can tag a variety of volumes, such as thin slabs or thin, pencil-like cylinders. The droplets in these volumes glow during a few Kolmogorov times. By tracking the fate of pencil-shaped clouds using a fast (5 kHz) camera, we come to the surprising conclusion that they disperse faster than fluid elements, with a spreading rate reaching a maximum at $\mathrm{St} \approx 2$. Sheets of tagged droplets display preferential concentration at work; we discuss statistical quantities that can capture these events.

[1]This project is funded by Fundamenteel Onderzoek der Materie (FOM)

11:48AM H14.00007 Wall Shear Stress in Oscillating Channel Flow Using Particle Image Velocimetry , BLAKE LANCE, Utah State University, JESSE ROBERTS, Sandia National Laboratories, BARTON SMITH, Utah State University, SEAN KEARNEY, Sandia National Laboratories — Offshore wind and water power are renewable sources with the potential for significant power generation. But each generation mechanism has risks from ocean floor structures that can disrupt natural sediment transport by increasing local shear stress. The Sediment Erosion Actuated by Wave Oscillations and Linear Flow (SEAWOLF) flume was designed and built to replicate wave motion with both oscillatory and unidirectional components to study sediment transport. The rectangular test section provides optical access for Particle Image Velocimetry (PIV) measurements. Additionally series of pressure taps allow for differential pressure measurements. Sine-wave oscillations and unidirectional flow in more than a dozen combinations are measured and presented. Phase locked measurements of volume flow rates, velocity fields, and pressure are acquired over several hundred cycles and phase averaged. High spatial resolution PIV is used near the wall for direct shear stress measurements. Since the flow is unsteady, the pressure drop in the test section has both inertial and friction contributions. To isolate the friction term, the pressure resulting from the fluid acceleration is subtracted. The synced PIV and pressure measurements on smooth walls where the viscous sublayer is formed confirm the accuracy of this method. The pressure sensor then measures shear stress on rough walls where the viscous sublayer is disrupted or non-existent and where optical access is difficult.

12:01PM H14.00008 The gas generation measurement at high electric filed in electrokinetic devices , MENA TAWFIK, THOMAS HANSEN, FRANCISCO J. DIEZ, Rutgers, The State University of New Jersey — In generating high EO flows, a limiting factor is faradaic reactions which appear at high electric fields. The gas released at the electrodes due to the faradic reaction forms bubbles which blocks the flow direction and increase the system resistance. To understand the factors that affect the bubble generated and its effect on the flow at high electric field, the gas generated is measured experimentally. Both DC and asymmetric bipolar rectangular voltage waveform are used to control the gas generated volume. Comparing the gas generated measured experimentally with that calculated theoretically, we found that the results have the same trend with about 50% gas generation efficiency. The gas generated volume is the same either applying DC or bipolar rectangular voltage waveform, however, it noticed that in the case of the bipolar rectangular voltage waveform due to the continuously switch of the flow direction, the bubble is detached early from the electrodes that leads to smaller bubbles formation compared to the DC voltage. The effect of the frequency of the bipolar rectangular voltage waveform is to reduce the gas generation by about 15% at high frequency >50 kHz. To the best of our knowledge this is the first time the gas generated was measured in electrokinetic devices.

12:14PM H14.00009 Mechanisms and methods for biofouling prevention via aeration , NATASHA DICKENSON, CHARLES HENOCH, JESSE BELDEN, Naval Undersea Warfare Center — Biofouling is a major problem for the Navy and marine industries, with significant economic and ecological consequences. Specifically, biofouling on immersed hull surfaces generates increased drag and thus requires increased fuel consumption to maintain speed. Considerable effort has been spent developing techniques to prevent and control biofouling, but with limited success. Control methods that have proven to be effective are costly, time consuming, or negatively affect the environment. Recently, aeration via bubble injection along submerged surfaces has been shown to achieve long-lasting antifouling effects, and is the only effective non-toxic method available [1,2]. An understanding of the basic mechanisms by which bubble-induced flow impedes biofouling is lacking, but is essential for the design of large-scale systems. We present results from an experimental investigation of several bubble induced flow fields over an inclined plate with simultaneous measurements of the fluid velocity and bubble characteristics using Digital article Image Velocimetry and high speed digital video. Trajectories of representative larval organisms are also resolved and linked with the flow field measurements to determine the mechanisms responsible for biofouling prevention.

[1] Scardino et al. 2009. *J Mar Sci Technol*. No. A13. pp.3-10.
[2] Bullard et al. 2010. *Aquatic Invasions*. 26: 587-593.

12:27PM H14.00010 Optical sensor for detection of supercavity-body contact location[1] , JESSE BELDEN, MICHAEL JANDRON, Naval Undersea Warfare Center, TADD TRUSCOTT, Brigham Young University — Supercavitating vehicles have been the subject of intense research due to the potential for drag reduction and/or increased speeds. The control of such vehicles depends on accurate knowledge of planing forces generated by partial, transient wetting of afterbody surfaces. Measurement of the supercavity-body contact location, which determines the planing area, is thus critical for vehicle control. A robust sensor capable of measuring supercavity contact location along the length of a body is presented. The sensor operates on the optical principle of total internal reflection to differentiate between liquid and gas phases in contact with the body. An array of photodetectors is used to sense the presence or absence of light from a laser source to map the contact location. The theoretical operation and limitations of the sensor are discussed and several experiments are presented to validate the theory. Also, we present an elegant signal processing method to compensate for *in situ* changes in ambient light conditions.

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

Monday, November 25, 2013 10:30AM - 11:35AM –

Session H16 Reacting Flows V: Kinetics 304 - Venkat Raman, University of Texas at Austin

10:30AM H16.00001 An Assessment of the RCCE for Computationally Efficient Combustion Simulations with Detailed Kinetics , FATEMEH HADI, Northeastern University, MOHAMMAD JANBOZORGI, University of Southern California, REZA H. SHEIKHI, HAMEED METGHALCHI, Northeastern University — The Rate-Controlled Constrained-Equilibrium (RCCE) method is assessed for detailed kinetics simulations in combustion. The method describes the reacting system dynamics by a relatively small number of rate-controlling reactions and slowly-varying constraints. The unconstrained chemical species are assumed to be in a temporary constrained-equilibrium state and their compositions are determined by maximizing the entropy. The RCCE is applied to predict methane combustion in a constant pressure Partially-Stirred Reactor (PaSR) using 12 constraints and 133 reaction steps. Simulations are carried out over a wide range of initial temperatures and equivalence ratios. The RCCE predictions are compared with those obtained from direct integration of detailed kinetics. It is demonstrated that the set of constraints chosen, accurately represents the methane oxidation kinetics. The effect of mixing on reaction is studied for different residence and mixing time scales. Results show that the RCCE provides accurate prediction of reaction dynamics with various levels of mixing. The RCCE is also shown to significantly reduce the stiffness and the overall computational cost associated with detailed kinetics.

10:43AM H16.00002 An adjoint approach for determining sensitivity of laminar flames , KALEN BRAMAN, The University of Texas at Austin, TODD OLIVER, Institute for Computational Engineering and Sciences, The University of Texas at Austin, VENKAT RAMAN, The University of Texas at Austin — Combustion simulations involve a large number of parameters including chemical rate coefficients and species diffusivities. When comparing such simulations to experimental data, it becomes essential to know the relative impact of each of these parameters on the target quantity. For problems that involve a small number of simulation targets and a large number of parameters, adjoint-based sensitivity analysis is highly efficient. In this work, we develop the continuous adjoint equations for a laminar flame configuration, and provide a numerical algorithm for the solution of the dual problem. Simulations of a hydrogen flame are used to test this new approach. Key results pertaining to model validation are discussed.

10:56AM H16.00003 The Quantum-Kinetic Chemical Reaction Model for Navier-Stokes Codes , MICHAEL A. GALLIS, ROSS M. WAGNILD, JOHN R. TORCZYNSKI, Sandia National Laboratories — The Quantum-Kinetic chemical reaction model of Bird is formulated as a non-equilibrium chemical reaction model for Navier-Stokes codes. The model is based solely on thermophysical, molecular-level information and is capable of reproducing measured equilibrium reaction rates without using any experimentally measured reaction-rate information. The model recognizes the principal role of vibrational energy in overcoming the reaction energy threshold. The effect of rotational non-equilibrium is introduced as a perturbation to the effect of vibrational non-equilibrium. Since the model uses only molecular-level properties, it is inherently able to predict reaction rates for arbitrary non-equilibrium conditions. This ability is demonstrated in the context of both Navier-Stokes and DSMC codes. 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.

11:09AM H16.00004 On the potential failure of reduced reaction kinetics , JOSEPH POWERS, SAMUEL PAOLUCCI, University of Notre Dame — Severe stiffness of equations modeling advection, reaction, and diffusion in combustion systems has motivated many efforts to filter the primary mechanism inducing the stiffness: the simultaneous presence of fast and slow reaction dynamics. Here, it is demonstrated that a common filtering technique for construction of low dimensional reaction manifolds, connection of equilibria by heteroclinic orbits, can fail. While the method is guaranteed to generate an invariant manifold, the local dynamics far from equilibrium may be such that nearby trajectories are in fact carried away from the identified invariant manifold, thus rendering it to be of limited utility in capturing slow dynamics far from equilibrium. An eigenvalue-based method is described to characterize the local behavior of such invariant manifolds. The method provides a diagnostic tool for evaluating whether a candidate manifold has the desirable properties of being both slow and attractive. A simple model system and a realistic hydrogen-air system are examined; method success and failure are demonstrated.

11:22AM H16.00005 Investigations of spontaneous ignition of high-pressure hydrogen release based on detailed chemical kinetics , HIROSHI TERASHIMA, MITSUO KOSHI, TOSHIO MOGI, RITSU DOBASHI, The University of Tokyo — A numerical simulation of spontaneous ignition of high-pressure release in a length of duct is performed to explore ignition mechanisms. The present study adopts a rectangular duct and focuses on effects of initial diaphragm shape on spontaneous ignitions. The Navier-Stokes equations with a detailed chemical kinetics mechanism are solved in a manner of direct numerical simulation. A conventional numerical approach is used for solving the Navier-Stokes equations, while the chemical source term is integrated by a dynamic multi-timescale method for alleviating the stiffness. Detailed mechanisms of spontaneous ignitions are discussed for various initial diaphragm shapes. For a straight diaphragm shape, the ignition occurs only near the wall region due to the adiabatic wall condition, while, for a largely deformed diaphragm shape, the three ignition events: ignition due to leading shock wave reflection at the wall, hydrogen penetration into shock-heated air near the wall, and deep penetration of hydrogen into shock-heated air behind the leading shock wave, are identified.

Monday, November 25, 2013 10:30AM - 12:27PM –

Session H17 Biofluids: Locomotion VI - Swimming and Flapping Models 305 - Oscar Curet, Florida Atlantic University

10:30AM H17.00001 Simple asymptotic results for the role of flexibility in flapping propulsion , MATTHEW N.J. MOORE, Courant Institute of Mathematical Sciences — Wing or fin flexibility in flapping propulsion is important to our understanding of bio-locomotion and may be used to engineer devices based on similar principles. Laboratory experiments and numerical simulations have been used extensively to examine effects of wing flexibility, but useful analytical results seem to be lacking. Here we use a small-amplitude calculation to determine the forces produced by a thin wing flapping in an inviscid, 2D fluid and shedding a vortex-sheet wake. We represent flexibility in a simple way by considering a torsional spring located at the root of a rigid wing. The wing moves according to an imposed heaving motion and pitches passively in response to the fluid and spring forces. Remarkably, closed-form expressions are obtained for the kinematics and thrust produced by the wing. Though limited to small amplitude, the results capture a variety of behaviors that are consistent with previous experimental and numerical observations. For small frequencies, thrust is enhanced by torsional compliance and peaks at a resonant frequency, while for larger frequencies the compliant wing underperforms when compared to a clamped, rigid wing. The wing can even produce negative thrust, i.e. drag, if the wing's mass is sufficiently large.

10:43AM H17.00002 Predicting Fruit Fly's Sensing Rate From Insect Flight Simulations , JANE WANG, SONG CHANG, Cornell University — Without sensory feedbacks, flies cannot fly. Exactly how sensory feedback controls work in flying insects is a complex puzzle to solve. What do insects measure in order to stabilize their flight? What kinds of neural computations and muscle activities are involved in order to correct their flight course or to turn? How often and how fast do animals adjust their wings to remain stable? To understand the algorithms used by insects to control their dynamic instability, we have developed a simulation tool to study flapping flight, where motions of the insect body and wings are coupled instantaneously. To stabilize the flight in the simulation, we construct a control algorithm that modulates wing motion based on discrete measurements of the body-pitch orientation. Our simulations give theoretical bounds both on the sensing rate and the delay time between sensing and actuation. Interpreting these findings together with experimental results on fruit flies' reaction time and sensory motor reflexes, we give a sharper bound on the sensing rate and further reason that fruit flies sense their kinematic states every wing-beat in order to stabilize their flight.

10:56AM H17.00003 Insect flight on fluid interfaces: a chaotic interfacial oscillator , HARIPRIYA MUKUNDARAJAN, MANU PRAKASH, Stanford University — Flight is critical to the dominance of insect species on our planet, with about 98 percent of insect species having wings. How complex flight control systems developed in insects is unknown, and arboreal or aquatic origins have been hypothesized. We examine the biomechanics of aquatic origins of flight. We recently reported discovery of a novel mode of "2D flight" in Galerucella beetles, which skim along an air-water interface using flapping wing flight. This unique flight mode is characterized by a balance between capillary forces from the interface and biomechanical forces exerted by the flapping wings. Complex interactions on the fluid interface form capillary wave trains behind the insect, and produce vertical oscillations at the surface due to non-linear forces arising from deformation of the fluid meniscus. We present both experimental observations of 2D flight kinematics and a dynamic model explaining the observed phenomena. Careful examination of this interaction predicts the chaotic nature of interfacial flight and takeoff from the interface into airborne flight. The role of wingbeat frequency, stroke plane angle and body angle in determining transition between interfacial and fully airborne flight is highlighted, shedding light on the aquatic theory of flight evolution.

11:09AM H17.00004 Does dragonfly's abdomen flexion help with fast turning maneuvers?[1] , GENG LIU, CHENGYU LI, HAIBO DONG, Dept. of Mechanical and Aerospace Engineering, University of Virginia, FLOW SIMULATION RESEARCH GROUP TEAM — Dragonflies are able to achieve fast turning maneuvers during take-off flights. Both asymmetric wing flapping and abdomen flexion have been observed during the fast turning. It's widely thought that the asymmetric wing beats are responsible of producing the aerodynamic moment needed for the body rotation. However, the dynamic effect of the abdomen flexion is not clear yet. In this study, an integrated experimental and computational approach is used to study the underlying dynamic effect of dragonfly abdomen flexion. It's found that dragonfly abdomen tended to bend towards the same side as the body reorienting to. Quantitative analysis have shown that during take-off turning maneuver the abdomen flexion can modulate the arm of force by changing the position of the center of mass relative to the thorax. As a result, roll and yaw moments produced by the wing flapping can be enhanced. This work is supported by NSF CBET-1313217.

[1]This work is supported by NSF CBET-1313217

11:22AM H17.00005 Swimming near deformable membranes at low Reynolds number[1] , MARCELO A. DIAS, THOMAS R. POWERS, Brown University — Microorganisms are rarely found in Nature swimming freely in an unbounded fluid. Instead, they typically encounter other organisms, hard walls, or deformable boundaries such as free interfaces or membranes. Hydrodynamic interactions between the swimmer and nearby objects lead to many interesting phenomena, such as changes in swimming speed, tendencies to accumulate or turn, and coordinated flagellar beating. Inspired by this class of problems, we investigate locomotion of microorganisms near deformable boundaries. We calculate the speed of an infinitely long swimmer close to a flexible surface separating two fluids; we also calculate the deformation and swimming speed of the flexible surface. When the viscosities on either side of the flexible interface differ, we find that fluid is pumped along or against the swimming direction, depending on which viscosity is greater.

[1]National Science Foundation Grant No. CBET-0854108

11:35AM H17.00006 On the interactions between two undulatory swimmers and between a swimmer and a boundary , JINZHOU YUAN, HAIM BAU, University of Pennsylvania — We study numerically and experimentally the interactions between a low-Reynolds number, undulatory swimmer, such as C. elegans, and a non-slip wall and the interactions between two swimmers in an unbounded domain. The Stokes equation with collision avoidance potential was solved using finite elements to obtain the translational and rotational drag coefficients of the swimmers. The swimmers' instantaneous linear and angular velocities were determined by requiring the swimmers to be subject to zero net forces and torques and using the method of superposition. A swimmer proximate to a wall is attracted to the wall and eventually assumes a trajectory that is parallel to the wall and a speed that is twice that of a comparable swimmer distal from the wall. The theoretical predictions are in qualitative agreement with experimental observations. Under certain circumstances, two swimmers in an unbounded domain attract one another and eventually achieve an equilibrium distance between their centers of mass and an equilibrium phase difference. The equilibrium distance between the swimmers and the phase difference between their gaits are functions of the swimmers' initial positions and orientations.

11:48AM H17.00007 Symmetry breaking of rigid/flexible plates in bluff body wakes generates rotation and drift , NICOLAS BROSSE, UGIS LACIS, FREDRIK LUNDELL, SHERVIN BAGHERI, Linne Flow Centre, KTH Mechanics, 10044 Stockholm, Sweden, FRANCOIS INGREMEAU, HAMID KELLAY, Centre de Physique Moleculaire Optique et Hertzienne (UMR 5798 CNRS), Universite Bordeaux I, 33405 Talence, France, ANDREA MAZZINO, INFN and CINFAI Consortium, Genova Section, Via Dodecaneso 33, 16146 Genova, Italy — Bluff body wakes have historically been important for understanding nature and aiding industry. For Reynolds numbers above approximately $Re \approx 10$, a recirculation bubble develops behind the bluff body. If a solid or elastic appendage is attached to the bluff body, it may exert a torque and a side force on the body. We use theory, numerical simulations and experiments to investigate and explain this phenomenon. More specifically, numerical simulations are carried out for a freely falling cylinder with an attached splitter plate for $Re \approx 50$. Experiments of a fixed cylinder with an attached elastic filament are preformed using a vertical soap-film tunnel for $Re \approx 2000$. Both experiments and simulations reveal that if a body has an appendage smaller than or of the same order as the body it is attached to, the body rotates and drifts. We explain our findings with a simple model and discuss the implications for propulsion.

12:01PM H17.00008 On the role of reduction by symmetry in understanding swimming at mid-Reynolds[1] , HENRY JACOBS, Imperial College London — A number of numerical and experimental studies suggest suggest that swimming can be characterized as an emergent phenomena arising from time-periodic internal body forces. In particular, it seems reasonable to surmise that swimming can be characterized as a relative limit cycle. A relative limit cycle is a system trajectory with a time-period, wherein each period is related to the previous by the action of a Lie group. In the case of swimming in R^n this Lie group is the set of rotations and translations, $\mathrm{SE}(n)$. In this talk we will describe a class of dissipative systems which admit relative limit cycles. Unfortunately, the Navier-Stokes equations coupled to a solids in R^n are not within this class of. However, a Navier-Stokes-α fluid on the n-sphere, S^n, could resolve this issue. The relative limit cycles would be with respect to the group $\mathrm{SO}(n)$. In a very precise sense, the group $\mathrm{SO}(n)$ is to the S^n as $\mathrm{SE}(n)$ is to R^n. As a result, the relative limit cycles obtained on S^n, can be characterized as spatially localized manifestations of trajectories for systems in R^n wherein each period related to the next by a rigid rotation and translation.

[1]H.O.J. is supported by European Research Council Advanced Grant 267382 FCCA

12:14PM H17.00009 Efficient kinematics for jet-propelled swimming , SILAS ALBEN, University of Michigan, LAURA MILLER, UNC-Chapel Hill, JIFENG PENG, U. Alaska-Fairbanks — We use vortex sheet and viscous simulations and an analytical model to search for efficient jet-propelled swimming kinematics at large Reynolds numbers (1000 and above). We prescribe different power-law kinematics for the bell contraction and expansion. In the simulations, two types of efficient kinematics are found: a bell radius velocity which is a nearly linear function of time, and a "burst-and-coast" kinematics. The analytical model studies the contraction phase only, and finds that the efficiency-optimizing kinematics transition from a nearly linear bell radius velocity (similar to the numerics) for small-to-moderate output power to an exponentially-decaying bell radius velocity for large output power.

Monday, November 25, 2013 10:30AM - 12:01PM –

Session H18 Biofluids: General IV - Plant Biomechanics 306/307 - Laura Miller, University of North Carolina

10:30AM H18.00001 Reconfiguration of broad leaves into cones , LAURA MILLER, University of North Carolina at Chapel Hill — Flexible plants, fungi, and sessile animals are thought to reconfigure in the wind and water to reduce the drag forces that act upon them. Simple mathematical models of a flexible beam immersed in a two-dimensional flow will also exhibit this behavior. What is less understood is how the mechanical properties of a leaf in a three-dimensional flow will passively allow roll up and reduce drag. This presentation will begin by examining how leaves roll up into drag reducing shapes in strong flow. The dynamics of the flow around the leaf of the wild ginger *Hexastylis arifolia* are described using particle image velocimetry. The flows around the leaves are compared with those of simplified sheets using 3D numerical simulations and physical models. For some reconfiguration shapes, large forces and oscillations due to strong vortex shedding are produced. In the actual leaf, a stable recirculation zone is formed within the wake of the reconfigured cone. In physical and numerical models that reconfigure into cones, a similar recirculation zone is observed with both rigid and flexible tethers. These results suggest that the three-dimensional cone structure in addition to flexibility is significant to both the reduction of vortex-induced vibrations and the forces experienced by the leaf.

10:43AM H18.00002 Physical Limits to Leaf Size in Tall Trees[1] , KAARE JENSEN, N. MICHELE HOLBROOK, Harvard University, MACIEJ ZWIENIECKI, University of California, Davis — Leaf size in angiosperm trees vary by more than three orders of magnitude, from a few mm to over 1 m. This large morphological freedom is, however, only expressed in small trees and the observed leaf size range declines with tree height, forming well-defined upper and lower boundaries. We recently showed (Phys. Rev. Lett. **110**, 018104 (2013)) that the limits to leaf size can be understood by physical constraints imposed by the microfluidic sugar transport network. The lower boundary is set by a minimum Péclet number, the upper boundary by a diminishing gain in transport efficiency.

[1]This work was supported by the Materials Research Science and Engineering Center at Harvard University (MRSEC, NSF Grant No. DMR- 0820484)

10:56AM H18.00003 Nuclear traffic and peloton formation in fungal networks , MARCUS ROPER, PATRICK HICKEY, STEPHANIE LEWKIEWICZ, UC Los Angeles, EMILIE DRESSAIRE, Trinity College, NICK READ, University of Manchester — Hyphae, the network of microfluidic pipes that make up a growing fungal cell, must balance their function as conduits for the transport of nuclei with other cellular functions including secretion and growth. Constant flow of nuclei may interfere with the protein traffic that enables other functions to be performed. Live-cell imaging reveals that nuclear flows are anti-congestive; that groups of nuclei flow faster than single nuclei, and that nuclei sweep through the colony in dense clumps. We call these clumps pelotons, after the term used to describe groups of cycle racers slip-streaming off each other. Because of the pelotons, individual hyphae transport nuclei only intermittently, producing long intervals in which hyphae can perform their other functions. Modeling reveals how pelotons are created by interactions between nuclei and the hyphal cytoskeleton, and reveal the control that the fungus enjoys over peloton assembly and timing.

11:09AM H18.00004 Moss hair water transport , ZHAO PAN, Brigham Young University, NAN WU, Key Laboratory of Biogeography and Bioresource in Arid Land, Chinese Academy of Sciences, RANDY HURD, SCOTT THOMSON, WILLIAM PITT, TADD TRUSCOTT, Brigham Young University — We present an investigation of water transportation on a moss (*Syntrichia caninervis*) indigenous to temperate deserts. The moss typically appears to be in a dry, brown state, but is rehydrated by water during the wet season, making the desert green. Small hairs (500-2000 μm in length, and 40 μm in diameter, d) growing out from the tip of the moss leaves transport water back to the leaves. Through high speed observations and mathematical modeling it appears that this transportation is driven by two different mechanisms. 1) Droplet transport is achieved in three ways: i) A large ($10d$) droplet attached between two intersecting fibers will move toward the bases of the leaves by the changing angle between the two hairs. ii) The shape of the moss hair is conical, thicker at the base, producing a gradient that moves fluid ($5d$) toward the leaf similar to cactus spines. iii) We also observe that in some cases a Plateau-Rayleigh instability trigger a series of droplets moving toward the base. 2) Micro-grooves on the moss hair transport a film of water along the moss hair when larger droplets are not available. These various water transportation strategies combine to help the moss to survive in the desert and provide valuable insight.

11:22AM H18.00005 Control of fluidic environments by mushrooms , EMILIE DRESSAIRE, JUNIUS SANTOSO, LISA YAMADA, Trinity College, MARCUS ROPER, UCLA — Thousands of fungal species rely on mushrooms for spore release and dispersal. Long distance spore dispersal by wind is instrumental to maintain genetic diversity and to the spread of pathogenic species. The conventional view is that fungi enjoy little control over the mechanism of dispersal. A spore falling from the mushroom cap can only hope to be picked up by a favorable airflow and carried away from the gap between the mushroom cap and the ground. We show that fungi actively manipulate their local fluidic environment by altering the buoyancy of the air surrounding the mushroom using a combination of water vapor and active cooling. This manipulation allows spore escape and dispersal from caps that may be spaced a few millimeters above the ground, or apart from each other. Through high speed videography, scaling analysis and indirect measurements, we reveal the fluid mechanics of spore escape, and how they are controlled by the biophysical properties of the mushroom.

11:35AM H18.00006 Self-burial mechanics of hygroscopically active awns[1] , WONJONG JUNG, Seoul National University, WONJUNG KIM, Sogang University, HO-YOUNG KIM, Seoul National University — We present the results of a combined experimental and theoretical investigation of the mechanics of self-burial of some plant seeds whose morphologies respond to humidity change of the surroundings. The seeds of Pelargonium species have hygroscopically active awns that play a critical role in the dispersal from the parent plant and burial in soil. While the awn uncoils to a linear shape in a highly humid condition, it recoils to a helical shape when dry. The rotation is driven by the structure of the cell walls that are comprised of cellulose microfibers aligned in a tilted helix. During uncoiling of the awn, the revolving tail generates thrust to burrow into soil, so that the seed is self-buried. We present the direct observation of the self-burial of the seed with the thrust into a soft substrate being measured at the same time. The elastica theory allows us to rationalize this botanical digging mechanics using the structural deformations of the hygroexpansive tissues.

[1]This work was supported by the Sogang University Research Grant of 2013 (201310009.01) and the National Research Foundation of Korea (grant no. 2012-008023)

11:48AM H18.00007 Physicochemical hydrodynamics of porous structures in vascular plants[1] , JEONGEUN RYU, SUNGSOOK AHN, SEUNG-GON KIM, Pohang University of Science and Technology (POSTECH), TAEJOO KIM, The Korea Atomic Energy Research Institute (KAERI), SANG JOON LEE, Pohang University of Science and Technology (POSTECH) — Transport of sap flow through xylem conduits of vascular plants has been considered as a passive process, because the xylem conduits are regarded as inert, dead wood. However, plants can actively regulate water transport using ion-mediated response for adapting to environmental changes. In order to understand the active regulation mechanism of physicochemical hydrodynamics of porous structures in vascular plants, the effects of specific ion types and their ionic ratios on the water transport were experimentally investigated under *in vivo* condition. Based on the experimental results, the principle of ionic effects will be explained through *in-vitro* comparative experiments and theoretical considerations.

[1]This work was supported by the National Research Foundation of Korea(NRF) grant funded by the Korea government (MSIP) (No. 2008-0061991).

Monday, November 25, 2013 10:30AM - 12:27PM –

Session H19 Biofluids: Cellular III - Computational Studies on Mechanical Properties of Cellular Flows 310/311 - Jonathan Freund, University of Illinois

10:30AM H19.00001 Lipid tubule growth by osmotic pressure , PADMINI RANGAMANI, University of California, Berkeley, DI ZHANG, University of Washington, GEORGE ORSTER, University of California, Berkeley, AMY SHEN, University of Washington — We present here a procedure for growing lipid tubules in vitro. This method allows us to grow tubules of consistent shape and structure and thus can be a useful tool for nano-engineering applications. There are three stages during the tubule growth process: initiation, elongation and termination. Balancing the forces that act on the tubule head shows that the growth of tubules during the elongation phase depends on the balance between osmotic pressure and the viscous drag exerted on the membrane from the substrate and the external fluid. Using a combination of mathematical modeling and experiment, we identify the key forces that control tubule growth during the elongation phase.

10:43AM H19.00002 Lattice Boltzmann simulations of leukocyte rolling and deformation in a three-dimensional shear flow , YE LUO, DEWEI QI, Western Michigan University, GUOWEI HE, LNM, Institute of Mechanics, Chinese Academy of Sciences, Beijing, 100080, China — Lattice Boltzmann simulation is used to simulate the motion of a leukocyte in fluid. The cell membrane is built by lattice spring model. The interaction between the fluid flow and the solid surface is treated by immersed boundary method. Stochastic Monte Carlo method is used to deal with receptor/ligand interaction. It is shown that the model can correctly predict the characteristic "stop-and-g" motion of rolling leukocytes. Effects of cell deformation, shear rates, bonding force, microvilli distribution on rolling are studied and compared with experiments.

10:56AM H19.00003 Effect of asymmetric deformation on capsule lateral migration[1] , STEPHANIE NIX, YOHSUKE IMAI, DAIKI MATSUNAGA, TAKUJI ISHIKAWA, TAKAMI YAMAGUCHI, Tohoku University — In a Stokes flow, lateral migration is the movement of a particle perpendicular to the flow direction due to the presence of a wall and/or shear gradient. Lateral migration has an effect on microscale flows in a number of fields. For example, in the cardiovascular system, the presence of a cell-free layer in blood vessels near the vessel wall is caused by the lateral migration away from the wall. In this study, we use the boundary integral method to investigate the wall-induced lateral migration of a capsule, which consists of a hyperelastic membrane enclosing an inner fluid. The boundary integral equation can be separated into two terms that represent contributions due to the capsule shape and wall. We find that the extent of the asymmetrical deformation of the capsule works to decrease the rate of migration perpendicular to the wall by up to 30% compared to the far-field analytical solution. Additionally, the effect of the asymmetrical deformation persists for distances up to ten times the capsule radius. Since the effect of asymmetrical deformation is only weakly dependent on the membrane properties, this type of analysis could be useful towards the understanding of lateral migration of other particles, such as drops and vesicles.

[1]Supported by JSPS Research Fellowships for Young Scientists

11:09AM H19.00004 Relaxation of deformed drops, vesicles, and cells , MIAO YU, JIA ZHANG, HAO LIN, Mechanical and Aerospace Engineering Department, Rutgers University, JEFFREY ZAHN, Biomedical Engineering Department, Rutgers University, WENCHANG TAN, Department of Mechanics and Engineering Sciences, Peking University — The deformation of drops, vesicles, and cells constitutes an important class of problems in chemical and biomedical engineering, and is often explored as a means to study interfacial dynamics and mechanical properties of the lipid membrane. Less attention has been paid to the relaxation process after the deforming mechanism is removed. In this work, analyses of such process are presented. A drop, vesicle or cell of spherical shape at rest is initially deformed into a spheroid. The relaxation process is then solved within the same theoretical framework in both small- and moderate-deformation limits. Different regimes are discovered. For sufficiently small initial deformations, the change in the membrane tension is a negligible higher-order effect for both vesicles and cells, and they behave identically to drops in the relaxation process. For moderate initial deformations, vesicle and cell relaxation is dominantly governed by the folding of undulations on the lipid membrane which differs from the behavior of a drop. Membrane properties, namely, membrane tension and bending rigidity, are the key parameters governing this dynamic process. A detailed comparison with experimental data for vesicles/cells is performed, and the results are presented and discussed.

11:22AM H19.00005 Flow-induced segregation in confined multicomponent suspensions: Effects of particle size and rigidity , MICHAEL GRAHAM, AMIT KUMAR, University of Wisconsin-Madison — The effects of particle size and rigidity on segregation in confined flow of binary suspensions of fluid-filled capsules are investigated in a model system resembling whole blood. We study this problem using a boundary integral method as well as with a master equation model that incorporates wall-induced migration and hydrodynamic pair collisions. Boundary integral results indicate that, in a mixture of large and small particles, the small particles marginate, while the large particles antimarginate. Here margination refers to localization of particles near walls, while antimargination refers to the opposite. In a mixture of particles with equal size and unequal stiffness, the stiffer particles marginate while the flexible ones antimarginate. The master equation model traces the origins of these behaviors to the size and rigidity dependence of the wall-induced migration velocity and of the cross-stream particle displacements in various types of collisions. Finally, a set of coupled non-local drift-diffusion equations is derived, providing further insights in terms of the drift and diffusion of various species.

11:35AM H19.00006 The wall traction induced by flowing red blood cells in model microvessels and its potential mechanotransduction[1] , JONATHAN FREUND, University of Illinois at Urbana-Champaign, JULIEN VERMOT, IGBMC, CNRS/INSERM/UdS — There is evidence in early embryonic development, even well before advective oxygen transport is important, that the presence of red bloods cells *per se* trigger essential steps of normal vascular development. For example, Lucitti *et al.* [*Development* **134**, 3317 (2007)] showed that sequestration of blood cells early in the development of a mouse, such that the hematocrit is reduced, suppresses normal vascular network development. Vascular development also provides a model for remodeling and angiogenesis. We consider the transient stresses associated with blood cells flowing in model microvessels of comparable diameter to those at early stages of development (6μm to 12μm). A detailed simulation tool is used to show that passing blood cells present a significant fluctuating traction signature on the vessel wall, well above the mean stresses. This is particularly pronounced for slow flows ($\lesssim 50\mu$m/s) or small diameters ($\lesssim 7\mu$m), for which root-mean-square wall traction fluctuations can exceed their mean. These events potentially present mechanotranduction triggers that direct development or remodeling. Attenuation of such fluctuating tractions by a viscoelastic endothelial glycocalyx layer is also considered.

[1]NSF supported

11:48AM H19.00007 Capturing mechanical properties of biological cells using coarse-grained modeling[1] , WENBIN MAO, MONIQUE CHANG, ALEXANDER ALEXEEV, Georgia Institute of Technology, Atlanta, Georgia — Understanding cell mechanics is important for a variety of biomedical applications. Our goal is to develop a coarse-grained computational model that can properly capture the micromechanics of biological cells. The coarse-grained cell model includes an elastic shell enclosing a cross-linked polymer network and a viscous fluid representing, respectively, cell membrane, cytoskeleton, and cytoplasm. We use this model to investigate the mechanical response of cells to external forces and compare the results with the experimental AFM measurements. We systematically vary the properties and structure of the internal polymer network and the outer membrane to assess their influence on the cell mechanical responses. This model not only reveals interesting insights into the cell mechanics, but also provides a promising tool for investigation of motile and multicellular systems.

[1]Acknowledge financial support from NSF under Award No. 0932510

12:01PM H19.00008 Effect of cytoskeleton stress-free state on red blood cell responses in low shear rate flows , QIANG ZHU, University of California, San Diego, ZHANGLI PENG, Massachussetts Insititute of Technology, ADEL MASHAYEKH, University of California, San Diego — Inspired by the recent experiment on erythrocytes (red blood cells, or RBCs) in weak shear flows (Dupire et al. 2012), we conduct a numerical investigation to study the dynamics of RBCs in low shear rate flows by applying a multiscale fluid-structure interaction model. By employing a spheroidal stress-free state in the cytoskeleton we are able to numerically predict an important feature that the cell maintains its biconcave shape during tank treading motions. This has not been achieved by any existing models. Furthermore, we numerically confirm the hypothesis that as the stress-free state approaches a sphere, the threshold shear rates corresponding to the establishment of tank treading decrease. By comparing with the experimental measurements, our study suggests that the stress-free state of RBCs is a spheroid which is close to a sphere, rather than a biconcave shape applied in existing models (the implication is that the RBC skeleton is prestressed in its natural biconcave state). It also suggests that the response of RBCs in low shear rate flows may provide a measure to quantitatively determine the distribution of shear stress in RBC cytoskeleton at the natural state.

12:14PM H19.00009 Nonlinear Response of Bio-Polymers Subject to Stretching Flow with Thermal Noise , MINGGE DENG, Brown University, LEOPOLD GRINBERG, T.J. Watson Research Center, BRUCE CASWELL, GEORGE KARNIADAKIS, Brown University — The dynamics of elastic filaments subject to hydrodynamic forces exhibits complex nonlinear dynamics in the neighborhood of stagnation points in the flow. Here, the motion of a single in-extensible bio-polymer with an-isotropic friction tensor subjected to a stretching flow is modeled with stochastic differential equations as well as dissipative particle dynamics simulations. Our results show that the negative tension induces a stretch-coil transition beyond a critical value, where the noise is amplificated due to the interaction between thermal noise and nonlinear effects.

Monday, November 25, 2013 10:30AM - 12:40PM –

Session H20 DFD/GPC Minisymposium: Global Climate Models: Dynamical Cores, Strengths and Weaknesses 315 - Jim Brasseur, Pennsylvania State University; Brad Marston, Brown University, John Wettlaufer, Oxford University and Yale University

10:30AM H20.00001 The spectral element dynamical core in the Community Atmosphere Model , MARK TAYLOR, Sandia National Laboratories — I will describe our work developing CAM-SE, a highly scalable version of the Community Atmosphere Model (CAM). CAM-SE solves the hydrostatic equations with a spectral element horizontal descritization and the hybrid coordinate Simmons & Burridge (1981) vertical discretization. It uses a mimetic formulation of spectral elements which preserves the adjoint and annihilator properties of the divergence, gradient and curl operations. These mimetic properties result in local conservation (to machine precision) of mass, tracer mass and (2D) potential vorticity, and semi-discrete conservation (exact with exact time-discretization) of total energy. Hyper-viscsoity is used for all numerical dissipation. The spectral element method naturally supports unstructured/variable resolution grids. We are using this capability to perform simulations with 1/8 degree resolution over the central U.S., transitioning to 1 degree over most of the globe. This is a numerically efficient way to study the resolution sensitivity of CAM's many subgrid parameterizations.

10:56AM H20.00002 A 3-D Finite-Volume Non-hydrostatic Icosahedral Model (NIM) , JIN LEE, NOAA/ESRL — The Nonhydrostatic Icosahedral Model (NIM) formulates the latest numerical innovation of the three-dimensional finite-volume control volume on the quasi-uniform icosahedral grid suitable for ultra-high resolution simulations. NIM's modeling goal is to improve numerical accuracy for weather and climate simulations as well as to utilize the state-of-art computing architecture such as massive parallel CPUs and GPUs to deliver routine high-resolution forecasts in timely manner. NIM uses innovations in model formulation similar to its hydrostatic version of the Flow-following Icosahedral Model (FIM) developed by Earth System Research Laboratory (ESRL) which has been tested and accepted for future use by the National Weather Service as part of their operational global prediction ensemble. Innovations from the FIM used in the NIM include: * A local coordinate system remapped spherical surface to plane for numerical accuracy (Lee and MacDonald, 2009), * Grid points in a table-driven horizontal loop that allow any horizontal point sequence (A.E. MacDonald, et al., 2010), * Flux-Corrected Transport formulated on finite-volume operators to maintain conservative positive definite transport (J.-L, Lee, ET. Al., 2010), * All differentials evaluated as finite-volume integrals around the cells, *Icosahedral grid optimization (Wang and Lee, 2011) NIM extends the two-dimensional finite-volume operators used in FIM into the three-dimensional finite-volume solvers designed to improve pressure gradient calculation and orographic precipitation over complex terrain. The NIM dynamical core has been successfully verified with various non-hydrostatic benchmark test cases such as warm bubble, density current, internal gravity wave, and mountain waves. Physical parameterizations have been incorporated into the NIM dynamic core and successfully tested with multimonth aqua-planet simulations. Recent results from NIM simulations will be presented at the Symposium.

11:22AM H20.00003 Dynamical cores and climate modeling[1] , PETER HJORT LAURITZEN, National Center for Atmospheric Research — In this talk an overview of the development of next generation dynamical cores in climate modeling is given. Fluid flow solvers intended for coupled climate system models must be designed to respect important physical properties related to conservation and the physical realizability of the computed solution. Demands for increased complexity and higher resolution has forced the modeling community to go back to the drawing board and develop highly scalable solvers on non-traditional spherical grids. In this talk an overview of these topics will be given with specific examples from NCAR's (National Center for Atmospheric Research) Community Atmosphere Model (CAM).

[1]sponsored by the National Science Foundation (NSF) and Department of Energy (DOE)

11:48AM H20.00004 Intercomparison of General Circulation Models for Hot Extrasolar Planet Atmospheres[1] , JAMES CHO, Queen Mary, University of London — In this collaborative work with I. Polichtchouk, C. Watkins, H. Th. Thrastarson, O. M. Umurhan, and M. de la Torre-Juárez, we compare five general circulation models (GCMs) which have been recently used to study hot extrasolar planet atmospheres (BOB, CAM, IGCM, MITgcm, and PEQMOD), under three test cases useful for assessing model convergence and accuracy. Such a broad, detailed intercomparison has not been performed thus far for extrasolar planets study. The models considered all solve the traditional primitive equations, but employ different numerical algorithms or grids (e.g., pseudospectral and finite volume, with the latter separately in longitude-latitude and "cubed-sphere" grids). The test cases are chosen to cleanly address specific aspects of the behaviors typically reported in hot extrasolar planet simulations: 1) steady-state, 2) nonlinearly evolving baroclinic wave, and 3) response to fast timescale thermal relaxation. When initialized with a steady jet, all models maintain the steadiness, as they should—except MITgcm in cubed-sphere grid. A very good agreement is obtained for a baroclinic wave evolving from an initial instability in spectral models (only). However, exact numerical convergence is still not achieved across the spectral models: amplitudes and phases are observably different. When subject to a typical "hot-Jupiter"-like forcing, all five models show quantitatively different behavior—although qualitatively similar, time-variable, quadrupole-dominated flows are produced. Hence, as have been advocated in several past studies, specific quantitative predictions (such as the location of large vortices and hot regions) by GCMs should be viewed with caution. Overall, in the tests considered here, spectral models in pressure coordinate (PEBOB and PEQMOD) perform the best and MITgcm in cubed-sphere grid performs the worst.

[1]This work has been supported by the Science and Technology Facilities Council, Westfield Small Grant, NASA Postdoctoral Program, and Institute for Theory and Computation, Harvard College Observatory.

12:14PM H20.00005 Exploring effects of different dynamical cores in global climate models on regional predictions[1] , CHRIS FOREST, Dept. of Meteorology & EESI, Pennsylvania State University — We investigate the uncertainty in regional climate response to patterns of sea surface temperature (SST) anomalies due to multiple sources including the choice of the dynamical core. We quantify the sensitivity of regional climate to localized SST anomaly perturbations via a global teleconnection operator (GTO) [2] (i.e., an empirical Green's function.) Structural uncertainty is sampled in two primary ways. (1) We use versions of the NCAR Community Atmospheric Model (CAM3.1, CAM4, and CAM5) to examine the dependence on the sub-grid scale physics parameterizations. (2) We vary the dynamical cores (spectral, finite volume, and HOMME) for each model. We focus on the seasonal climate response over extensive continental regions as well as global scales. Overall, we can explore the dependence of the GTO on physics parameterizations, model resolution, and dynamical cores and identify regions related to atmospheric circulation patterns that exhibit different response characteristics. We note that initial condition uncertainties require sufficient sample sizes to identify such dependencies.

[1]This work has been supported by DOE grant DE-SC-0005399 and was done in collaboration with Dr. Joseph J. Barsugli and Dr. Wei Li.
[2]Li et al. (2012), *J. Geophys. Res.*, **117**, D20103, doi:10.1029/2011JD017186

Monday, November 25, 2013 10:30AM - 12:14PM –

Session H21 Turbulence: Simulations IV - DNS Application 316 - Abhilash Chandy, University of Akron

10:30AM H21.00001 Direct Numerical Simulations of Turbulent Ekman Layers with Increasing Static Stability: Modifications to the Bulk Structure and Second-Order Statistics[1] , STIMIT SHAH, ELIE BOU-ZEID, Princeton University — Turbulent Ekman layers, with increasing static stability and Reynolds number, are studied using Direct Numerical Simulations. The highest stability under which continuous turbulence can be sustained is shown to be Reynolds number dependent. The highest Reynolds number flow displays a well-developed inertial range and a logarithmic layer, which is found to obey the Monin-Obukhov similarity theory under non-neutral conditions. The analyses then focus on the budgets of turbulent kinetic energy (TKE), vertical velocity variance, momentum and buoyancy fluxes, and temperature variance. Results indicate that, due to imposed stability, there is damping of vertical motions that leads to a reduction in the turbulent transport of Reynolds stress towards the wall. This reduced transport results in lower production of TKE, which is shown to be more significant than direct buoyant destruction in reducing TKE levels in stable conditions. The reduction in the vertical velocity variance results in significant drops in the production terms in the other second order budgets we study as well. Building on these findings, we conclude by illustrating that the vertical velocity variance is a better parameter to base vertical eddy-diffusivity and viscosity models on than the full TKE.

[1]Stimit Shah and Elie Bou-Zeid are funded by NSF under AGS-1026636

10:43AM H21.00002 Energy dynamics in turbulence generated through concentrated regions of intense kinetic energy[1] , AGUSTIN MAQUI, DIEGO DONZIS, Texas A&M University — The photo-dissociation of molecules produced by lasers has the capability of ejecting fragments with extremely high velocities, thus creating concentrated regions of very large momentum. It is of fundamental as well as practical interest to determine whether this concentrated momentum is sufficient to generate realistic turbulence. Incompressible direct numerical simulations (DNS) with concentrated sources along "lines" are used to represent the photo-dissociation of molecules. The numerical challenges associated with the implementation of strong gradients are presented in a detailed convergence study. A thorough analysis is performed on the different terms of acceleration that determine the evolution of the flow. Our results indicate that pressure and the convective acceleration redistribute most of the momentum both radially and among the components of acceleration. Radial statistics of the different components of velocity, gradients, and accelerations are also related to the time development of the flow and correlated with the eventual emergence of fully developed turbulence. Further results and consequences for particular cases realizable in laboratories will be discussed.

[1]The authors gratefully acknowledge the support of AFOSR.

10:56AM H21.00003 Modeling of the Gecko's skin microfibrillar structures using the Immersed Boundary method via DNS , ISNARDO ARENAS, KENNETH CARRASQUILLO, U. of Puerto Rico, STEFANO LEONARDI, U. of Texas Dallas, GUILLERMO ARAYA, FAZLE HUSSAIN, LUCIANO CASTILLO, Texas Tech U. — There is a current interest in surfaces that mimic the skin of some species (i.e., sharks, dolphins and geckos) in order to achieve drag reduction. The surface considered is based on the microfribrillar structures of a gecko's skin (Aksak et al. 2008). The structures are modeled by means of the immersed boundary method proposed by Fadlun et al. (2000). Direct simulations are performed to predict flow dynamics with a Reynolds number of 7000 based on the height of the channel and centerline velocity. The ratio of the height of the structure with respect to the height of the channel is approximately 0.05. The main motivation is to study how the microfribillar structures affect the momentum transfer from the viscous layer to the outer layer. The surface shows a reduction of the area affected by the shear stress due to the cavities formed by the pattern. As expected, the cavities create a low velocity zone thus decreasing the Reynolds shear stresses. Lambda-2 and Q-criterion were implemented to identify the elongated streamwise vortices. The results show that when compared to a flat channel the microfribillar structures tend to preserve these streamwise vortices instead of bursting into the outer layer which is a source of drag increase.

11:09AM H21.00004 DNS of stably stratified Taylor-Green vortex , ABBAS RAHIMI, ABHILASH J. CHANDY, Department of Mechanical Engineering, University of Akron, Akron, OH - 44325-3903 — Stratified flows, flows where density varies in one direction, have wide applications in some of the phenomena occurring in the atmospheric and ocean. Direct numerical simulations (DNS) of transition to turbulence in a stably stratified Boussinesq fluid are presented for the three-dimensional Taylor-Green vortex problem at different stratification and turbulence intensities measured in terms of different Froude (Fr) (∞ and $10^{-2} - 10^{-1}$) and Reynolds numbers (Re) (800 and 1600), respectively. Features investigated include temporal variations of the energy spectrum cascade, local Froude numbers, vertical shearing of the velocities, and dissipation of kinetic and potential energy. The results from these simulations demonstrated forward cascade of energy for high Re and revealed the strong anisotropic structure of turbulence and suppression of vertical motion under stratification.

11:22AM H21.00005 DNS of helicity-induced stratified turbulent flow , ABHILASH J. CHANDY, ABBAS RAHIMI, Department of Mechanical Engineering, University of Akron, Akron, OH - 44325-3903 — Helical flows undergoing density stratification have wide applications in meteorological phenomena such as dust devils, tornadoes, and hurricanes due to the complexity and disasters caused by them. Direct numerical simulations (DNS) of transition to turbulence in a stably stratified Boussinesq fluid are presented for different rotation and stratification intensities. In order to understand the effect of velocity on the energy cascade, comparisons are made between helicity initiated and non-helical flows. Results show that stratification decelerates the helicity decay and causes velocity and vorticity to align with each other. With respect to the helical and non-helical flow comparisons, the total energy in the presence of stratification decays faster with helicity. In addition, the behavior of length scales were examined by comparing temporal variations of the vertical shearing of velocities. Results showed a growing asymmetry with time in the case of helical flow, while non-helical flow stayed close to begin symmetric.

11:35AM H21.00006 New DNS and modeling results for turbulent pipe flow , ARNE JOHANSSON, GEORGE EL KHOURY, OLOF GRUNDESTAM, PHILIPP SCHLATTER, GEERT BRETHOUWER, KTH, LINNE FLOW CENTRE TEAM — The near-wall region of turbulent pipe and channel flows (as well as zero-pressure gradient boundary layers) have been shown to exhibit a very high degree of similarity in terms of all statistical moments and many other features, while even the mean velocity profile in the two cases exhibits significant differences between in the outer region. The wake part of the profile, i.e. the deviation from the log-law, in the outer region is of substantially larger amplitude in pipe flow as compared to channel flow (although weaker than in boundary layer flow). This intriguing feature has been well known but has no simple explanation. Model predictions typically give identical results for the two flows. We have analyzed a new set of DNS for pipe and channel flows (el Khoury et al. 2013, Flow, Turbulence and Combustion) for friction Reynolds numbers up to 1000 and made comparing calculations with differential Reynolds stress models (DRSM). We have strong indications that the key factor behind the difference in mean velocity in the outer region can be coupled to differences in the turbulent diffusion in this region. This is also supported by DRSM results, where interesting differences are seen depending on the sophistication of modeling the turbulent diffusion coefficient.

11:48AM H21.00007 Direct numerical simulation of turbulence in a bent pipe , PHILIPP SCHLATTER, AZAD NOORANI, KTH Mechanics, Stockholm, Sweden — A series of direct numerical simulations of turbulent flow in a bent pipe is presented. The setup employs periodic (cyclic) boundary conditions in the axial direction, leading to a nominally infinitely long pipe. The discretisation is based on the high-order spectral element method, using the code Nek5000. Four different curvatures, defined as the ratio between pipe radius and coil radius, are considered: $\kappa = 0$ (straight), 0.01 (mild curvature), 0.1 and 0.3 (strong curvature), at bulk Reynolds numbers of up to 11700 (corresponding to $Re_\tau = 360$ in the straight pipe case). The result show the turbulence-reducing effect of the curvature (similar to rotation), leading close to relaminarisation in the inner side; the outer side, however, remains fully turbulent. Prpoer orthogonal decomposition (POD) is used to extract the dominant modes, in an effort to explain low-frequency switching of sides inside the pipe. A number of additional interesting features are explored, which include sub-straight and sub-laminar drag for specific choices of curvature and Reynolds number: In particular the case with sub-laminar drag is investigated further, and our analysis shows the existence of a spanwise wave in the bent pipe, which in fact leads to lower overall pressure drop.

12:01PM H21.00008 Enstrophy along particle trajectories through vortex clusters in DNS of turbulent channel flow[1] , JASON HACKL, JAVIER JIMENEZ, Universidad Politecnica de Madrid — We augment the traditional study of wall-bounded turbulence from the Eulerian point of view by analyzing the Lagrangian trajectories of fluid tracers tracked in a DNS of a turbulent channel at $Re_\tau = 2000$. After storing consecutive fields for 50 wall units in time, ensembles of $O(10^6)$ particles seeded on short detached vortex clusters centered at both $y^+ \approx 200$ and the core of the channel are tracked backward ($T_b^+ = -50$) in time, then restarted forward. Velocity gradients are interpolated along trajectories for these particles for a total duration of 100 units ($T^+_{forward} = 50$ past the seeding instant), providing representative histories of enstrophy acquisition and loss by fluid particles throughout the expected lifetime of intense vortical structures. The statistics of initial position $X(T_b^+ = -50)$, along with joint and conditional statistics of temporal increments of velocity and vorticity throughout the complete simulation (from $T^+ = -50$ to 50), describe how the structures above the buffer layer, typically educed from Eulerian variables, act on fluid, clarifying our understanding. The corresponding results for particles initialized in the core are compared to the particles initialized around vortices centered at $y^+ = 200$.

[1]Supported by ERC MULTIFLOW and BSC-RES FI-2013-1-0028

Monday, November 25, 2013 10:30AM - 12:27PM –
Session H22 Turbulent Mixing II 317 - Mark Kimber, University of Pittsburgh

10:30AM H22.00001 Measurements of scalar probability density functions and conditional expectations[1] , AMIR BEHNAMIAN, STAVROS TAVOULARIS, University of Ottawa — High resolution, multi-sensor, hot/cold-wire measurements were made in passively heated, uniformly sheared turbulence in a wind-tunnel, with focus on terms in the scalar PDF equation that require closure models. For the homogeneous scalar field that was produced by a uniform mean temperature gradient, results conformed with the literature: the scalar PDF was essentially Gaussian; the conditional expectations of velocities upon the scalar value were approximately linear; and the conditional expectation of the scalar dissipation rate upon the scalar value was mildly anisotropic and had a shape that was similar to those of any of its three parts, which justifies the use of the streamwise part as a surrogate for the total. All these properties behaved very differently in two inhomogeneous scalar fields, namely a thermal mixing layer and the plume of a heated line source: the scalar PDF were distinctly sub-Gaussian; the conditional velocity expectations were non-linear functions of the scalar value; and the conditional scalar dissipation rates were very strongly anisotropic, as well as depending on the scalar value in fashions that differed strongly from those of any of their three parts.

[1]Supported by NSERC

10:43AM H22.00002 Revisiting measurements of small scale temperature fluctuations , CHRISTIAN GEBAUER, CARLA BAHRI, GILAD ARWATZ, YUYANG FAN, MARCUS HULTMARK, Princeton University — It is well known that high frequency temperature measurements are attenuated due to a non-flat frequency response. Based on the temperature correction model proposed by Arwatz et. al (under review), new experimental data is compared with existing measurements. Focus is laid on structure functions, probability density functions, and the behavior of small scale temperature fluctuations. Additionally, a new temperature sensor developed at Princeton University is utilized for further improvement of temperature measurements. The effect of temporal resolution on the temperature spectrum is investigated by comparing uncorrected data to corrected data and data acquired with the new fast response temperature sensor.

10:56AM H22.00003 Inverse cascades sustained by the transfer rate of angular momentum in a 3D turbulent flow , JAVIER BURGUETE, MIGUEL LOPEZ-CABALLERO, University of Navarra — The existence of energy cascades as signatures of conserved magnitudes is one of the universal characteristics of turbulent flows. In this work we present the evidence of an inverse cascade in a fully developed 3D experimental turbulent flow where the conserved magnitude is the angular momentum. We analyze the behavior of a fluid in a closed cavity where two inhomogeneous and strongly turbulent flows collide in a thin region. The experimental volume is a closed cylinder (diameter of 20cm) where two impellers rotate in opposite directions. A key characteristic of this setup the high stability of the propellers (the instantaneous fluctuations are below 0.1%). We have performed PIV and LDA measurements of the velocity fields. Typical characteristics of the turbulent flow in this setup are: turbulence intensity 50%, the $Re_\lambda = 900$, the Taylor microscale $\lambda_T = 1.8$mm and the integral scale $L_I = 15$mm. The analysis of the data series reveal that below the injection scales an inverse cascade can be identified (-1/3 in time, -7/3 in space) that can be explained as the transfer of angular momentum between the diferent fluid layers. A. de la Torre, J. Burguete, Phys Rev Lett 99 (2007) 054101. M. Lopez-Caballero, J. Burguete, Phys Rev Lett 110 (2013) 124501.

11:09AM H22.00004 The horizontal planar structure of kinetic energy in a model vertical-axis wind turbine array[1] , ANNA CRAIG, ROBERT ZELLER, FRANCISCO ZARAMA, JOEL WEITZMAN, Stanford University, JOHN DABIRI, California Institute of Technology, JEFFREY KOSEFF, Stanford University — Recent studies have indicated that arrays of vertical axis wind turbines (VAWTs) could potentially harvest significantly more power per unit land area than arrays composed of conventional horizontal axis wind turbines. However, to design VAWT arrays for optimal power conversion, a more comprehensive understanding of inter-turbine energy transfer is needed. In the presented study, a geometrically scaled array of rotating circular cylinders is used to model a VAWT array. The horizontal inter-cylinder mean fluid velocities and Reynolds stresses are measured on several cross-sections using 2D particle image velocimetry in a flume. Two orientations of the array relative to the incoming flow are tested. The results indicate that cylinder rotation drives asymmetric mean flow patterns within and above the array, resulting in non-uniform distributions of turbulent kinetic energy. The variability is observed to be directly related to the ratio of the cylinder rotation speed to the streamwise water velocity. Emphasis is placed on the implications of the asymmetries for power production.

[1]Work supported by a Stanford Graduate Fellowship to A.E.C, by funding to J.O.D. from ONR N000141211047 and the Gordon and Betty Moore Foundation through Grant GBMF2645, and by funding from the Environmental Fluid Mechanics Laboratory, Stanford University

11:22AM H22.00005 Anisotropy tensor invariant assessment for counter-rotating wind turbine wakes[1] , NICHOLAS HAMILTON, RAÚL BAYOÁN CAL, Portland State University — Model wind turbine arrays were tested in a suite of wind tunnel experiments to determine the wake-to-wake interaction and mixing for different counter-rotation schemes of turbine rotors. All configurations were comprised of a standard Cartesian arrangement (4×3) of turbines. A uniform rotation scheme formed the control against which were tested row-by-row, column-by-column, and checkerboard counter-rotation configurations. Stereo PIV measurements were made immediately upstream and downstream of both entrance and exit row turbines in the center of the wind tunnel. The full Reynolds stress anisotropy tensor, a_{ij}, was calculated for all measurement locations showing effects of sense of rotation of rotor blades on turbulent stresses. The invariants of the anisotropy tensor were calculated and compared further demonstrating the effects of rotation and further characterizing the turbulence within a wind turbine canopy layer. Results have implications on return-to-isotropy models used in wind turbine array simulations.

[1]National Science Foundation: ECCS-1032647

11:35AM H22.00006 Variation of the slope of the velocity power spectrum and intermittency factor corresponding to $160 < \mathrm{Re}_\lambda < 490$[1] , ALEJANDRO PUGA[2], University of California, Irvine — It has been observed in many studies, at sufficiently high Reynolds number and for an intermediate range of wavelengths that $E_{11}(\kappa) \sim \kappa^{-n}$. Kolmogorov's 1st and 2nd hypotheses state that n should approach 5/3 as the Reynolds number tends to infinity and that the variation of n from 5/3 is due to intermittency in the dissipation. Wind tunnel experiments are conducted where high intensity turbulence is generated by means of an active turbulence grid modeled after Makita's 1991 design (Makita, 1991) as implemented by Mydlarski and Warhaft (M&W, 1998). The goal of this study is to document the variation of n over a range of Re_λ from 160 to 490. The corresponding values of n are 1.46 and 1.55 where $n = 5/3 - 3.23Re_\lambda^{-0.56}$. This is in disagreement with Mydlarski and Warhaft who found that $n = 5/3 - 5.25Re_\lambda^{-2/3}$. The intermittency factor, μ, is obtained from the slope of the dissipation spectrum where $E_{11}^{\varepsilon}(\kappa) \sim \kappa^{\mu-1}$ and its variation is determined. The intermittency factor is calculated using the spatial derivative of the downstream velocity as determined from the temporal derivative and Taylor's hypothesis. As turbulence intensity increases, it had been hypothesized that μ would become zero. However, Sreenivasan and Kailasnath (S&K, 1992), in agreement with Praskovsky and Oncley (P&O, 1994), have found that μ appears to be nearly a constant of 0.25 ± 0.05. In the current study it is found that the intermittency exponent is nearly a constant, in agreement with Sreenivasan and Kailasnath, but has a value of 0.7.

[1]University of California, Irvine Research Fund
[2]Tim Koster, John C. LaRue

11:48AM H22.00007 Turbulent velocity and concentration measurements in a macro-scale multi-inlet vortex nanoprecipitation reactor , ZHENPING LIU, RODNEY FOX, JAMES HILL, MICHAEL OLSEN, Iowa State University — Flash Nanoprecipitation (FNP) is a technique to produce monodisperse functional nanoparticles. Microscale multi-inlet vortex reactors (MIVR) have been effectively applied to FNP due to their ability to provide rapid mixing and flexibility of inlet flow conditions. A scaled-up MIVR could potentially generate large quantities of functional nanoparticles, giving FNP wider applicability in industry. In the presented research, the turbulent velocity field inside a scaled-up, macroscale MIVR is measured by particle image velocimetry (PIV). Within the reactor, velocity is measured using both two-dimensional and stereoscopic PIV at two Reynolds numbers (3500 and 8750) based on the flow at each inlet. Data have been collected at numerous locations in the inlet channels, the reaction chamber, and the reactor outlet. Mean velocity and Reynolds stresses have been obtained based on 5000 instantaneous velocity realizations at each measurement location. The turbulent mixing process has also been investigated with passive scalar planar laser-induced fluorescence and simultaneous PIV/PLIF. Velocity and concentration results are compared to results from previous experiments in a microscale MIVR. Scaled profiles of turbulent quantities are similar to those previously found in the microscale MIVR.

12:01PM H22.00008 Characterization of the Flow Field Over an Ablative Surface[1] , MICHAEL ALLARD, CHRISTOPHER WHITE, University of New Hampshire, YVES DUBIEF, University of Vermont — Experiments are performed in a small-scale wind tunnel to investigate the complex coupling between an erodible surface and an eroding agent. The flow configuration is a spatially developing heated boundary layer flow over an ablative surface. Several variations of the inlet conditions, both for flow and temperature, are used to study the temporal and spatial development of ablation driven by coherent structures, such as vortices, and the response of turbulence to wall recession and emergence of roughness (ablation patterns). Characterization and comparison of velocity and thermal fields over ablative and non-ablative surfaces are reported in addition to qualitative observations of ablation patterns for vortex driven, laminar, and turbulent flow over an ablative surface.

[1]This work is supported by the NSF (CBET-0967224)

12:14PM H22.00009 Turbulent dispersivity under conditions relevant to airborne disease transmission between laboratory animals , SIOBHAN HALLORAN, WILLIAM RISTENPART, Dept. Chemical Engineering & Materials Science, University of California, Davis — Virologists and other researchers who test pathogens for airborne disease transmissibility often place a test animal downstream from an inoculated animal and later determine whether the test animal became infected. Despite the crucial role of the airflow in pathogen transmission between the animals, to date the infectious disease community has paid little attention to the effect of airspeed or turbulent intensity on the probability of transmission. Here we present measurements of the turbulent dispersivity under conditions relevant to experimental tests of airborne disease transmissibility between laboratory animals. We used time lapse photography to visualize the downstream transport and turbulent dispersion of smoke particulates released from a point source downstream of an axial fan, thus mimicking the release and transport of expiratory aerosols exhaled by an inoculated animal. We show that for fan-generated turbulence the plume width is invariant with the mean airspeed and, close to the point source, increases linearly with downstream position. Importantly, the turbulent dispersivity is insensitive to the presence of meshes placed downstream from the point source, indicating that the fan length scale dictates the turbulent intensity and corresponding dispersivity.

Monday, November 25, 2013 10:30AM - 12:40PM –

Session H23 DFD Minisymposium: Frontiers in Combustion Physics I 318 - Forman Williams, University of California - San Diego

10:30AM H23.00001 Direct Numerical Simulation of Turbulent Premixed Hydrogen/Air Flames in Sheared Turbulence and in Counterflow with Product Stratification , J.H. CHEN, Sandia National Labs — Petascale direct numerical simulations (DNS) have been performed of canonical turbulent configurations to glean physical insight into turbulence-chemistry interactions in combustion and to provide validation data for the development of coarse-grained models for engineering CFD. The role of DNS is illustrated through two examples. In the first example, DNS of turbulent hydrogen/air premixed flames interacting with intense shear driven turbulence in the thin reaction zones regime at turbulent Reynolds numbers approaching 1000 (Hawkes et al. 2012) are performed over a range of Damköhler numbers. The DNS data are used to study inter-scale energy transfer through one-dimensional spectra of turbulent kinetic energy and reactive scalars from the turbulent premixed flames. Balance equations for the density weighted turbulent kinetic energy and scalar fluctuation spectra for reacting flows are derived and used to understand the physical processes unique to reacting flows. In the second example, DNS of highly turbulent lean premixed hydrogen-air flames stabilized against counterflowing non-adiabatic stoichiometric combustion products in chemical equilibrium are performed. The influence of product stratification on the mechanisms associated with local extinction and re-ignition in turbulent stratified combustion is studied.

In collaboration with H. Kolla, Sandia National Labs; A. Kerstein, 72 Lomitas Road Danville, CA 94526; N. Swaminathan, Cambridge University; E.R. Hawkes, University of New South Wales; S. Lyra, B. Coriton, J.H. Frank, Sandia National Labs; and A. Gomez, Yale University.

10:56AM H23.00002 The know unknowns: Detailed simulations and low-order modeling to characterize facility-induced non-idealities in chemical-kinetics experiments , MATTHIAS IHME, Stanford University — Experimental investigations to study chemical-kinetics processes, reaction-rates or ignition properties are frequently accompanied by facility-induced non-idealities. Examples are turbulence and thermo-viscous boundary layers in rapid compression machines, temperature fluctuations and mixture inhomogeneities in flow-reactors, or shock-bifurcations and pressure drifts in shock-tubes. Although experimental investigations are carefully conducted to mitigate these effects, they are difficult to quantify experimentally. Simulations can assist in identifying these non-idealities and in guiding experimental instrumentation to improve measurement accuracies. This presentation discusses three different modeling approaches to characterize facility-effects in rapid compression machines, flow reactors, and shock-tubes. After providing an overview about these facilities and describing the underlying models, examples are presented to illustrate effects of turbulence, mixture-inhomogeneities, heat-losses, and thermal stratification on the ignition dynamics in these facilities. Diagnostics is developed to assess the sensitivity of the induction chemistry and to quantify reliable operating regimes that are not contaminated by these non-ideal processes.

11:22AM H23.00003 Spray combustion: scales, regimes, and formulations[1] , ANTONIO L. SÁNCHEZ, Departamento de Ingeniería Térmica y de Fluidos, Universidad Carlos III de Madrid, Leganés 28911, Spain — This talk will cover some recent results relevant to the modeling of spray flames. Controlling parameters and combustion regimes will be reviewed. Conditions will be identified under which analyses of laminar mixing layers can shed light on aspects of turbulent spray combustion. Conservation equations will be derived for dilute sprays, including separate equations for the gas and liquid phases. Linear combinations of the gas-phase conservation equations for the species and energy will be used to formulate the problem in terms of chemistry-free coupling functions, including the relevant mixture fraction and the total enthalpy, which are not conserved scalars, because their conservation equations include source terms associated with the vaporizing droplets. Implications for spray-flamelet modeling, associated with the multivalued spatial dependence of the mixture fraction, will be explained. Applications of the coupling-function formulation to the computation of spray flames in the limit of infinitely fast reaction rate will be discussed, including the high-order corrections needed to account for the presence of droplets on the air side of the flame. Recent work on ignition of spray flames will also be presened.

In collaboration with Daniel Martínez-Ruiz, Departamento de Ingeniería Térmica y de Fluidos, Universidad Carlos III de Madrid, Leganés 28911, Spain; Javier Urzay, Center for Turbulence Research, Stanford University, Stanford, CA, USA; and Amable Liñán, ETSI Aeronáuticos, Pl. Cardenal Cisneros 3, Madrid 28040, Spain.

[1]This work was supported by the Spanish MCINN through project CSD2010-00011.

11:48AM H23.00004 Modeling Interactions Among Turbulence, Gas-Phase Chemistry, Soot and Radiation Using Transported PDF Methods , DANIEL HAWORTH, The Pennsylvania State University — The importance of explicitly accounting for the effects of unresolved turbulent fluctuations in Reynolds-averaged and large-eddy simulations of chemically reacting turbulent flows is increasingly recognized. Transported probability density function (PDF) methods have emerged as one of the most promising modeling approaches for this purpose. In particular, PDF methods provide an elegant and effective resolution to the closure problems that arise from averaging or filtering terms that correspond to nonlinear point processes, including chemical reaction source terms and radiative emission. PDF methods traditionally have been associated with studies of turbulence-chemistry interactions in laboratory-scale, atmospheric-pressure, nonluminous, statistically stationary nonpremixed turbulent flames; and Lagrangian particle-based Monte Carlo numerical algorithms have been the predominant method for solving modeled PDF transport equations. Recent advances and trends in PDF methods are reviewed and discussed. These include advances in particle-based algorithms, alternatives to particle-based algorithms (e.g., Eulerian field methods), treatment of combustion regimes beyond low-to-moderate-Damköhler-number nonpremixed systems (e.g., premixed flamelets), extensions to include radiation heat transfer and multiphase systems (e.g., soot and fuel sprays), and the use of PDF methods as the basis for subfilter-scale modeling in large-eddy simulation. Examples are provided that illustrate the utility and effectiveness of PDF methods for physics discovery and for applications to practical combustion systems. These include comparisons of results obtained using the PDF method with those from models that neglect unresolved turbulent fluctuations in composition and temperature in the averaged or filtered chemical source terms and/or the radiation heat transfer source terms. In this way, the effects of turbulence-chemistry-radiation interactions can be isolated and quantified.

12:14PM H23.00005 Investigation of turbulent spherical flames , N. SWAMINATHAN, Cambridge University Engineering Department — The role of turbulence is generally taken to be the main cause for the growth of flame-brush thickness in turbulent spherical flames and Taylor's dispersion theory had been used in past studies to support this. Contrary to this view, this study shows that the differential propagation between the leading and trailing edges of the flame-brush is the predominant cause for the growth of the flame-brush thickness with time in the spherical flames. The leading edge accelerates continuously because of the cumulative effect of flow acceleration resulting from heat release. These insights are derived by analysing URANS computations of 7 spherical and 7 planar flames having combustion conditions in the corrugated flamelets and thin reaction zones regimes. The reaction rate closure is achieved using strained premixed flamelets with scalar dissipation rate as a parameter. Detailed analyses of the results showed that the mean reaction rate does not depend on the flame geometry, planar or spherical. However, the turbulent flame speed which is the leading edge displacement speed showed a flame geometry dependence due to the geometry dependence of turbulent scalar flux. The presentation will highlight these physical insights.

In collaboration with I. Ahmed, Cambridge University Engineering Department.

Monday, November 25, 2013 10:30AM - 12:40PM –

Session H24 Aerodynamics III 319 - Jana Anirban, Pittsburgh Supercomputing Center

10:30AM H24.00001 An Experimental Investigation on the Interferences among Multiple Turbines in Onshore and Offshore Wind Farms[1] , WEI TIAN, AHMET OZBAY, HUI HU, Iowa State University — We report an experimental study to investigate the wake interferences among multiple wind turbines on onshore and offshore wind farms. The experimental studies are conducted in a large-scale Aerodynamic/Atmospheric Boundary Layer (AABL) Wind Tunnel with an array of scaled three-blade Horizontal Axial Wind Turbine (HAWT) models placed in atmospheric boundary layer winds with different mean and turbulence characteristics to simulate the situations in onshore and offshore wind farms. In addition to measuring dynamic wind loads (both forces and moments) and the power outputs of the scaled turbine models, a Particle Image Velocity (PIV) system is used to conduct detailed flow field measurements to quantify the turbulent wake vortex flows and the wake interferences among the wind turbines sited over onshore and offshore wind farms with non-homogenous surface winds. The detailed flow field measurements are correlated with the dynamic wind loads and power output measurements to elucidate underlying physics in order to gain further insight into the characteristics of the dynamic wind loads and wake interferences among multiple wind turbines for higher total power yield and better durability of the wind turbines.

[1]The research work is funded by NSF and IAWIND.

10:43AM H24.00002 Energetic Turbulence Structures in the Wake of Model Wind Turbines , JIAN SHENG, FARAZ MEHDI, Texas Tech University, LEONARDO P. CHAMORRO, University of Illinois at Urbana-Champaign — Wind turbine wakes contain complex and energetic flow structures. Characterizing the near-wake field is critical to assess flow-structure interactions and evaluate asymmetric loadings that trigger premature structural failure. Although the turbulence flow structure in the far-wake region is important in the wind farm design, an integrated characterization of the entire wake flow would provide clearer mechanistic view on other phenomena such wake meandering and unsteady interactions with the blades of downwind turbines. High-speed Particle Image Velocimetry (PIV) is carried out over a model wind turbine in a neutrally stratified boundary layer flow. The measurements are made at consecutive locations ranging from three rotor diameters upstream to twelve rotor diameters downstream of the unit. Vortical structures within the wake including tip, root and hub vortices are identified and followed as they advect downstream. The evolution of these dominant near-wake flow structures are quantified and provide us a better understanding of interactions between turbine wake and boundary layer. The spatial distribution of the mean and fluctuating velocity, as well as energy spectrum and turbulent kinetic budget are also discussed.

10:56AM H24.00003 Computational analysis of a tip vortex structure shed from a bio-inspired blade[1] , SEBASTIAN GOMEZ, LINDSAY N. GILKEY, BRYAN E. KAISER, SVETLANA V. POROSEVA, University of New Mexico — Understanding and predicting a tip vortex structure and its dynamics is of significant importance for all branches of aerodynamics. A particular focus of our research is the rotorcraft performance which is substantially influenced by a tip vortex. A tip vortex also is a major source of energy losses and acoustic noise. In the present study, an impact of a blade shape on a tip vortex structure is analyzed. Simulations are conducted of flows around a rectangular blade and a bio-inspired blade of the same area. An insect wing is chosen as a blade prototype. Indeed, insects developed physical characteristics that reduce energy consumption while permitting sustained and controlled flight at low level of noise. Analysis has been done to determine what insect poses flight characteristics closest to the small rotorcraft design goals. Commercial CFD software STAR-CCM+ is used for conducting computations on structured and unstructured grids and for data post-processing.

[1]The authors acknowledge support from UNM CARC in a form of access to HPC and from CD-Adapco for providing Star-CCM+ for academic purposes. The first author's work was supported by the New Mexico Space Grant Consortium.

11:09AM H24.00004 Analysis of Wake Profiles for Free Leading Edge Membranes in Low Reynolds Number Flow[1] , ANDREW WRIST[2], ZHENG ZHANG[3], JAMES P. HUBNER[4], University of Alabama — MAVs (micro air vehicles) are similar in size and flight velocity to nature's evolved flyers such as bats. Bats have flexible membrane wings that provide them with aerodynamic advantages, effectively reducing energy necessary to maintain flight. This study was inspired by the free LE (leading edge) and TE (trailing edge) combinations found on certain bat species. In previous research, silicone substitutes for these membranes have been tested on rigid frames, and it was found in certain cases that their lift-to-drag ratios outperform those of flat plates. In this study, wake profiles for different LE/TE combinations were analyzed, as increasing wake depth and width are related to increasing drag. Silicone membranes with an aspect ratio of one were constructed and tested at various angles of attack, pretensions, and fixed/free LE/TE configurations in a low speed wind tunnel at 10 m/s (Re = 50,000). The wake of each membrane configuration was measured using a hotwire probe. The results indicate that membrane airfoils with free LEs produced a greater momentum deficit due to increased losses on the leeward side of the membrane. Further characteristics and trends are discussed in the presentation.

[1]University of Alabama NSF REU Grant #1062611.
[2]Undergraduate research assistant
[3]Graduate research assistant
[4]Associate professor

11:22AM H24.00005 Aerodynamic Design of Wing based on Humpback Whale Flipper , SAIF AKRAM, FAISAL BAIG, Aligarh Muslim University — The tubercles provide a bio-inspired design that has commercial viability for wing-like structures. Wind tunnel tests at low speeds of model humpback flippers with leading-edge tubercles have demonstrated improvements tubercles make, such as a staggering 32% reduction in drag, 8% improvement in lift, and a 40% increase in angle of attack over smooth flippers before stalling. The tubercles on the leading edge act as a passive-flow control device that improves the performance and maneuverability of the flipper. Possible fluid-dynamic mechanisms for improved performance include delay of stall through generation of a vortex and modification of the boundary layer, and increase in effective span by reduction of both spanwise flow and strength of the tip vortex. In the present work, numerical investigation of a 3D wing with scalloped leading edge inspired by the humpback whale flipper is carried out at high subsonic speeds with variation in angle of attack from 0 to 25 degrees. The effect of using different turbulence models is also investigated in order to attain a better understanding of mechanism(s) responsible for improved aerodynamic performance. This new understanding of humpback whale flipper aerodynamics has strong implications for wing design.

11:35AM H24.00006 ABSTRACT WITHDRAWN –

11:48AM H24.00007 Flexible body with drag independent of the flow velocity , THOMAS BAROIS, EMMANUEL DE LANGRE, LadHyX, Ecole Polytechnique, MECHANICS AND LIVING SYSTEMS TEAM — The drag of a rigid object is expected to increase with flow velocity. For wide ranges of velocities commonly found, the drag increases as the square of the relative velocity of the fluid. This strong dependence of the load with velocity accounts for specific survival strategies adopted by passive living systems such as plants in wind or algae in marine environments: through elastic reconfiguration, the drag on plants is reduced when compared to a rigid configuration and the velocity exponent for the drag is typically found between 1 and 1.5. In this work, a flexible body conceptual model is presented that exhibits a drag force that is almost independent of the free stream velocity. This surprising result is shown to be remarkably robust as it is experimentally observed for a range of geometries. This study opens the way for the design of devices subjected to a drag that is independent of the flow velocity. This possibility constitutes a key point in various fields involving flexible structures that are towed or subjected to wind.

12:01PM H24.00008 Large-eddy simulations of the Lillgrund wind farm , KARL NILSSON, KTH Mechanics, SIMON-PHILIPPE BRETON, Uppsala University Campus Gotland, STEFAN IVANELL, DAN HENNINGSON, KTH Mechanics — Large-eddy simulations using the EllipSys3D Navier-Stokes solver developed at DTU/Risø combined with an actuator disc (ACD) method for rotor modeling are performed to compute the power production of the turbines in the Lillgrund wind farm. The ACD method models the rotor with body forces determined from drag and lift coefficients which are tabulated as functions of the angle of attack. As the boundary layer over the blades is not resolved, this approach greatly reduces the computational costs compared to simulations involving the modeling of the full blade geometry. The simulations are performed both with a recently implemented power controller, which forces the turbines to adapt their rotational speed to the conditions they are operating in, and without any controller, where all turbines are given a fixed rotational speed. The atmospheric conditions are modeled using pre-generated turbulence and a prescribed boundary layer. Only a marginal difference is found between the results from the simulations with and without the controller. The simulation results show a very good agreement with measured production from the real farm. Therefore, it can be concluded that the simulation method realistically predicts the power production of the Lillgrund wind farm.

12:14PM H24.00009 Experimental study on influence of pitch motion on the wake of a floating wind turbine model , STANISLAV ROCKEL, ForWind - University of Oldenburg, Germany, RAUL BAYOAN CAL, Department of Mechanical and Materials Engineering, Portland State University, OR, JOACHIM PEINKE, MICHAEL HOELLING, ForWind - University of Oldenburg, Germany — Wind energy has become a major contributor to energy from renewable sources and is still demanded to increase its portion to the overall energy supply. Offshore wind energy was found to have the highest potential to fulfill these demands, due to better and steadier wind conditions found on seas. Offshore wind turbines which have been installed lately use monopiles as foundations and are feasible in shallow water up to a depth of 50m. Such shallow areas are rare and often exploited, so floating support structures for offshore wind turbines in deep water are possible solutions. The additional degrees of freedom of a floating support structure will influence the aerodynamics at the rotor and its wake. Wind tunnel experiments were performed using a classical fixed turbine model and a streamwise oscillating turbine in free pitch motion. For both cases the turbines were operated under same inflow conditions and wakes up to 7 rotor diameters were measured using 2D-3C stereographic particle image velocimetry (SPIV). The obtained data was statistically analyzed and a direct comparison of the wake of the fixed and oscillating turbine was performed. Our results show that inclinations and oscillations of the turbine have a strong impact on the structure of the wake and its development.

12:27PM H24.00010 ABSTRACT WITHDRAWN –

Monday, November 25, 2013 10:30AM - 12:27PM –

Session H25 Flow Control V: Injection/Suction 320 - Michael Amitay, Rensselaer Polytechnic Institute

10:30AM H25.00001 Aerodynamic Control of a Pitching Airfoil by Distributed Bleed Actuation[1] , JOHN KEARNEY, ARI GLEZER, Georgia Institute of Technology — The aerodynamic forces and moments on a dynamically pitching 2-D airfoil model are controlled in wind tunnel experiments using distributed active bleed. Bleed flow on the suction surface downstream of the leading edge is driven by pressure differences across the airfoil and is regulated by low-power louver actuators. The bleed interacts with cross flows to effect time-dependent variations of the vorticity flux and thereby alters the local flow attachment, resulting in significant changes in pre- and post-stall lift and pitching moment (over 50% increase in baseline post-stall lift). The flow field over the airfoil is measured using high-speed (2000 fps) PIV, resolving the dynamics and characteristic time-scales of production and advection of vorticity concentrations that are associated with transient variations in the aerodynamic forces and moments. In particular, it is shown that the actuation improves the lift hysteresis and pitch stability during the oscillatory pitching by altering the evolution of the dynamic stall vortex and the ensuing flow attachment during the downstroke.

[1]Supported by the Rotorcraft Center (VLRCOE) at Georgia Tech.

10:43AM H25.00002 The effects of local blowing perturbations on thermal turbulent structures , CAN LIU, GUILLERMO ARAYA, Texas Tech U., STEFANO LEONARDI, U. of Texas Dallas, LUCIANO CASTILLO, Texas Tech U. — Blowing is an active flow control technique with several industrial applications, particularly in film cooling of turbine blades. In the past, the effects of localized blowing have been mostly analyzed on the velocity field and its influence of the flow parameters and turbulence structures (Krogstad and Kourakine, 2000). However, little literature can be found on the effects of blowing on the coherent thermal structures. In the present study, an incompressible turbulent channel flow with given steady blowing at the wall is simulated via DNS by means of five spanwise holes. The Reynolds number based on the friction velocity and half channel height is approximately Re = 394 and the molecular Prandtl number is Pr = 0.71. Temperature is considered a passive scalar with isothermal conditions at the wall. Different blowing amplitudes and perturbing angles (with respect to the streamwise direction) are applied to find out their effects on the turbulent thermal structures by means of a two-point correlation analysis. In addition, local reduction and increase of drag are connected to vorticity. The corresponding influence of perturbing amplitudes and angles on the energy budget of thermal fluctuations and turbulent Prandtl numbers are also shown and discussed.

10:56AM H25.00003 A Linear Proportional Control of Turbulent Flow in a Planar Asymmetric Diffuser[1] , DONGGUN SON, HAECHEON CHOI, Seoul National University — We perform a linear proportional control of turbulent flow in a planar asymmetric diffuser (Obi diffuser) for separation delay and pressure recovery. The Reynolds number based on the half of inlet channel height (δ) and bulk mean velocity (u_b) is $Re_b = u_b\delta/\nu = 9000$, which is the same condition as done by previous experimental and numerical studies. An actuation for the control is defined at the diffuser throat ($x/\delta = 0$ to 1) as a wall-normal blowing and suction. A sensing variable (error) for the control is the difference between the instantaneous wall shear stresses at the upper and lower walls. The linear proportional control successfully suppresses the separation bubble at the lower slant wall and reduces the skin friction at the upper flat wall, resulting in the pressure recovery at the exit of diffuser. At an optimal proportional gain, the present control produces 6.7% increase in the exit pressure with delayed separation.

[1]Supported by the NRF Program (NRF-2011-0028032, NRF-2012M2A8A4055647)

11:09AM H25.00004 Wavelet diagnostics of the flow control of unsteady separation on a 2D Wind Turbine Airfoil[1] , ZHE BAI, JACQUES LEWALLE, GUANNAN WANG, MARK GLAUSER, Syracuse University — We investigated the aerodynamic characteristics of a 2D wind turbine airfoil. Unsteadiness was associated with the wake of a cylinder upstream of the airfoil. The experiments were conducted in both the baseline case, and with active closed-loop control on the suction surface of the airfoil. The data consisted of surface pressure time series. Continuous wavelet analysis gave the phase, band-pass filtered signals and envelope of harmonics of the fundamental shedding frequency. Coherence of pairs of signals was also used to map the flow characteristics. For the baseline and controlled case, we will report on the relation between phase of the leading edge fluctuations, unsteady flow separation and lift and drag coefficients. Our goal is to develop a more effective controller.

[1]The experiment was funded by DoE through University of Minnesota Wind Energy Consortium. Thanks for the support from the MAE department of Syracuse University.

11:22AM H25.00005 The Role of Vorticity Injection in Separation Control[1] , KUNIHIKO TAIRA, PHILLIP MUNDAY, Florida State University — Large eddy simulation is performed to examine the role of vorticity injection in separation control of spanwise periodic flow over a NACA0012 airfoil. The computations are conducted with a high-fidelity LES solver CharLES with sufficient grid resolution to resolve the near-wall turbulence at a moderate Reynolds number of $Re = 23,000$. The actuator input is introduced to the flow field through the velocity boundary condition to specify the desired vorticity flux input. The aim of this investigation is to analyze the influence of the injected vorticity magnitude and direction on the separation physics over the airfoil such that the separation is delayed. The vortical perturbation is added to break apart the large spanwise vortices responsible for causing separation and hence delay stall. The range of the vorticity injected is chosen to match those from commonly used flow control devices for separation control. In this study, particular focus is placed on examining the interaction between the actuator input and the inherent Kelvin-Helmholtz and spanwise instabilities.

[1]Work supported by AFOSR (Award No. FA9550-13-1-0183).

11:35AM H25.00006 Numerical Investigation of Virtual Aeroshaping Due to Pitched Synthetic Jets , JASON LI, ONKAR SAHNI, MANE, RPI — Synthetic jets in a (non-separated) crossflow provide a virtual aeroshaping effect in a time-averaged sense, which alters the local characteristics of the crossflow such as pressure gradient. The ability to manipulate virtual aeroshaping is beneficial, e.g., jets mounted on an aerial vehicle allow for control of aerodynamic behavior. A numerical investigation is conducted to study the effects of a pitched synthetic jet actuator on virtual aeroshaping, where the resulting recirculation zones behind the actuators are analyzed. Both geometric and operational parameters of actuators are varied in simulations based on permutations of one geometric parameter: pitch angle (60°, 75°, 90°), and two operational parameters: blowing ratio (to be in O(1.0-2.0)), and actuation frequency (to be in O(500-1000Hz)). In these simulations, the jet is placed in a laminar crossflow (e.g., Blasius boundary layer over a flat plate). A stabilized finite element method with implicit time integration technique is employed.

11:48AM H25.00007 Dynamic Stall of Finite Span Blades and its Control , KEITH TAYLOR, CHIA LEONG, MICHAEL AMITAY, Rensselaer Polytechnic Institute — An experimental investigational study into a dynamically pitching s809 airfoil at a Reynolds number of 220,000 was conducted. Particle Image Velocimetry was employed to visualize and quantify the flow field around the airfoil. This investigation compares a 2-D configuration with 3-D configuration (i.e., a finite span blade). The difference in the flow field between these two configurations is explored, as the vibrations present in the 3-D configuration (due to the dynamic stall) may contribute to a different apparent flow field than classical results would suggest. In addition, a comparison between lift and drag coefficients, measured on the 2-D and 3-D configurations, is explored, demonstrating how time varying lift and drag forces oscillate at characteristic frequencies associated with the primary vibrational modes of the model. In addition, flow control is applied through the actuation of an array of synthetic jets located near the leading edge of the model, in order to effect changes in the flow field around the model, demonstrating how dynamic stall can be delayed or eliminated during dynamic conditions.

12:01PM H25.00008 Active Flow Control of a Transonic Shock over Curved Surfaces , ABRAHAM N. GISSEN, BOJAN VUKASINOVIC, ARI GLEZER, Georgia Institute of Technology, SIVARAM P. GOGINENI, Spectral Energies — The effects of fluidic actuation on the evolution and dynamics of a transonic shock over a two-dimensional convex surface by controlling the ensuing shock-induced separation are investigated in wind tunnel experiments. Actuation is effected by a spanwise array of high-frequency (nominally 10 kHz) fluidic oscillating jets. The flow field upstream and downstream of the shock is investigated using high-speed Schlieren and PIV (3,000fps), and surface pressure measurements. It is shown that control of the shock-induced separating shear layer by exploiting direct control of small-scale motion can alter the degree of flow attachment and have a profound effect on the shock dynamics. The actuation diminishes shock oscillations near the surface, and leads to streamwise shock displacement that is proportional to the actuation strength (as measured, for example, by the mass flow rate coefficient). The strong correlation between the shock displacement and surface pressure are explored for application of closed-loop control.

12:14PM H25.00009 3-D Separation Control using Spatially-Compact, Pulsed Actuation[1] , GEORGE T.K. WOO, ARI GLEZER, Georgia Institute of Technology — The dynamics of controlled 3-D transitory attachment of stalled flow over a dynamically pitching 2-D airfoil are investigated in wind tunnel experiments. Pulsed actuation is effected over a spanwise fraction of the separated domain on a time scale that is an order of magnitude shorter than the airfoil's characteristic convective time scale using surface-integrated pulsed, combustion-driven actuator jets. The formation, evolution, and advection of vorticity concentrations over the airfoil and in its near wake are computed from high-resolution, phase-locked PIV measurements of the flow field in multiple cross-stream planes. It is shown that transitory attachment spreads toward the outboard, unactuated flow domains and exceeds the spanwise width of the actuation. The attachment is preceded by the formation of 3-D vortical structures that are advected and shed into the near wake. The effect of the actuation on the variation of the lift and pitching moment during the pitching cycle is altered significantly with its phase delay relative to the airfoil's pitching motion and can significantly mitigate the adverse aerodynamic effects of the dynamic stall.

[1]Supported by AFOSR.

Monday, November 25, 2013 10:30AM - 12:14PM –

Session H26 Reacting Flows VI: Premixed 321 - Guillaume Blanquart, California Institute of Technology

10:30AM H26.00001 High Karlovitz n-Alkane Premixed Flame DNS: Effects of Turbulence on the Flame Structure , BRUNO SAVARD, BROCK BOBBITT, GUILLAUME BLANQUART, California Institute of Technology — The effects of turbulence on the structure of a statistically flat, slightly lean ($\phi = 0.9$), n-heptane/air premixed flame is investigated using three dimensional direct numerical simulations at a Karlovitz number close to 100. Two simulations are performed: one with unity Lewis numbers and one with non-unity Lewis numbers. The first simulation is used to investigate deviations away from the laminar flamelet structure as eddies penetrate the preheat and reaction zones. The second is to analyze the relative importance of molecular vs turbulent mixing and their effects on species transport. The conditional mean profiles of the species mass fraction vs temperature from both simulations are evaluated to assess the influence of turbulence on scalar transport. As expected, turbulent mixing and molecular mixing are of comparable magnitude and, as a result, the structure of the flame is altered. Using a method developed in a previous work, the effective Lewis numbers of the different species are identified. These effective Lewis numbers are closer to unity than their laminar value, showing the effect of turbulent mixing. Interestingly, with this change in Lewis numbers, the structure of the turbulent flame compares very favorably with that of a laminar flame.

10:43AM H26.00002 High Karlovitz *n*-Alkane Premixed Flame DNS: Effects of the Flame on Turbulence Characteristics , BROCK BOBBITT, BRUNO SAVARD, GUILLAUME BLANQUART, California Institute of Technology — The study of premixed turbulent combustion requires understanding turbulence and chemistry independently as well as their effects upon one another. This coupling alters their inherent characteristics in a complex fashion. Unfortunately, the transformation of the turbulence across the flame is not well understood and it is common to assume homogeneous, isotropic turbulence before and after the flame. To this end, direct numerical simulations were performed of homogeneous isotropic turbulence interacting with a premixed flame. These were done at a Karlovitz number of approximately 100 using both tabulated and detailed *n*-heptane air chemistry. The integral length scale was four times the laminar flame thickness allowing study of both large and small scale turbulence. The transformation of these turbulent scales across the flame was investigated throughout and behind the flame. A model for the transfer function was developed by applying a generalized expansion to the continuity, momentum, vorticity, temperature, and species transport equations. From this, equations are derived which describe to leading order the transformation of turbulent velocity and length scales across the flame.

10:56AM H26.00003 A simulation of a bluff-body stabilized turbulent premixed flame using LES-PDF , JEONGLAE KIM, STEPHEN POPE, Cornell University — A turbulent premixed flame stabilized by a triangular cylinder as a flame-holder is simulated. The computational condition matches the Volvo experiments (Sjunnesson *et al.* 1992). Propane is premixed at a fuel lean condition of $\phi = 0.65$. For this reactive simulation, LES-PDF formulation is used, similar to Yang *et al.* (2012). The evolution of Lagrangian particles is simulated by solving stochastic differential equations modeling transport of the composition PDF. Mixing is modeled by the modified IEM model (Viswanathan *et al.* 2011). Chemical reactions are calculated by ISAT and for the good load balancing, PURAN distribution of ISAT tables is applied (Hiremath *et al.* 2012). To calculate resolved density, the two-way coupling (Popov & Pope 2013) is applied, solving a transport equation of resolved specific volume to reduce statistical noise. A baseline calculation shows a good agreement with the experimental measurements in turbulence statistics, temperature, and minor species mass fractions. Chemical reaction does not significantly contribute to the overall computational cost, in contrast to non-premixed flame simulations (Hiremath *et al.* 2013), presumably due to the restricted manifold of the purely premixed flame in the composition space.

11:09AM H26.00004 Impact of Chemistry Models on Flame-Vortex Interaction , SIMON LAPOINTE, BROCK BOBBITT, GUILLAUME BLANQUART, California Institute of Technology — In premixed turbulent combustion, accurate modeling of the fuel chemistry for numerical simulations is a critical component of capturing the relevant physics. Various chemical models are currently being used including detailed chemistry, tabulated chemistry, one-step chemistry, and rate-controlled constrained-equilibrium. However, the differences between these models and their impacts on the fluid dynamics are not well understood. Towards that end, the interaction between a laminar premixed hydrogen flame and a two-dimensional vortex was studied through Direct Numerical Simulations using each of these different chemistry models. In these simulations, the flame thickness, flame speed, viscosity, diffusivity, conductivity, density ratio, and vortex characteristics were held constant providing comparison of the effects of each chemical model alone. A convergence study was performed for each model assessing the numerical requirements of domain size, grid spacing and time step to completely resolve both the fluid dynamics and the chemistry. The converged results from each model were compared by considering the evolution of the flame structure and characteristics of the vortex.

11:22AM H26.00005 Folds and pockets in the propagation of premixed turbulent flames , NAVIN FOGLA, MOSHE MATALON, University of Illinois at Urbana Champaign, FRANCESCO CRETA, University of Rome La Sapienza — We examine the propagation of premixed flames in two-dimensional turbulent flows within the context of a hydrodynamic model that treats the flame as a surface of density discontinuity using a hybrid Navier-Stokes/interface capturing technique. Employing an improved interface capturing technique, which allows the flame front to attain multivalued configurations and form pockets of unburned gas before being consumed, broadens the range of applicability of our results to include the corrugated flamelet regime ($u'/S_L > 1$) of turbulent combustion. Three regimes are identified, depending on the mixture composition, thermal expansion coefficient and turbulence intensity: a regime where, on the average, the flame brush remains planar and unaffected by the Darrieus-Landau (DL) instability, a regime where the DL effects, responsible for frequent intrusions of the flame front into the burned gas region, have a marked influence on the flame brush that remains resilient to turbulence, and a highly turbulent regime where the influences of the DL instability progressively decrease and play limited to no role on the flame propagation. Particular attention is given in this presentation to the effects of folding/pocket formation on the flame structure and dynamics.

11:35AM H26.00006 Turbulent combustion modeling using explicit convolution of 1-D laminar flame , S. MUKHOPADHYAY, R.J.M. BASTIAANS, J.A. VAN OIJEN, L.P.H. DE GOEY, Technische Universiteit Eindhoven — Increasing computational power is enabling highly resolved Large Eddy Simulation (LES) of turbulent reacting flows. However resolving chemical scales in a practical combustor even with tabulated chemistry methods, still remains unaffordable and requires a model. DNS of a premixed slot flame is performed and *a priori* analysis indicates that laminar flame filtered at suitable scale can represent the chemical state in a turbulent reacting flow. But to represent all the chemical states, multiple filter widths will be required. This work explores a new modeling approach, Filtered Flamelet Generated Manifold (FFGM) based on explicit convolution of 1-D laminar flame solutions with spatial filter kernel of varying widths. To test the validity of the model *a posteriori* analysis, using tabulated chemistry constructed by convoluting a premixed laminar flame with top hat kernel of multiple widths is performed for the DNS configuration. The results indicate good performance of the model compared to DNS at a fraction of computational cost.

11:48AM H26.00007 Numerical forcing of an M-flame: linear analysis , MATHIEU BLANCHARD, LadHyX - Ecole Polytechnique, PETER J. SCHMID, LadHyX, CNRS - Ecole Polytechnique, DENIS SIPP, ONERA, THIERRY SCHULLER, SEBASTIEN CANDEL, EM2C, CNRS and Ecole Centrale Paris — Direct numerical simulations of a high Mach number (Ma=0.1), low Reynolds number (Re = 1000), premixed, lean M-flame have been studied with the goal of characterizing and quantifying the response of this generic flame to acoustic modulations. This response is essential to a description of thermo-acoustic instabilities. The flame is submitted to energy disturbances introduced in the injection tube of reactants using random binary signals. The unit impulse response of the flow variables in the burnt gases is computed. It features disturbances of acoustic and hydrodynamic nature. The short time response of this function is controlled by acoustic disturbances, while large hydrodynamic perturbations dominate the long time response of the unit impulse function. The mechanisms controlling the short and long time responses of the flame are examined. A sensitivity analysis is then conducted for selected characteristic frequencies and the structures of the linear optimal forcing are determined.

12:01PM H26.00008 Flame Thickness and Conditional Scalar Dissipation Rate in a Premixed Temporal Turbulent Reacting Jet , SWETAPROVO CHAUDHURI, Indian Institute of Science, HEMANTH KOLLA, Sandia National Laboratories, EVATT HAWKES, The University of New South Wales, JACQUELINE CHEN, Sandia National Laboratories, CHUNG LAW, Princeton University — The flame structure corresponding to a lean H_2/air premixed flame in intense sheared turbulence in the thin reaction zones regime is quantified from flame thickness and conditional scalar dissipation rate statistics obtained from recent direct numerical simulation data of premixed temporally-evolving turbulent slot jet flames. The local alignment between the progress variable iso-surface normals and the most compressive principal strain rate is observed to increase traversing from the unburnt reactants to the burnt products. Such preferential alignment coupled with sub-unity Le associated with lean H_2/air mixtures results in increasing normalized mean conditional scalar dissipation rate and a resultant decrease of the normalized mean flame thickness. On average, these turbulent flames are thinner than their corresponding planar laminar flames. The intermittency of the conditional scalar dissipation rate is found to exhibit a unique non-monotonicity of the exponent of the stretched exponential function, conventionally used to describe probability density function tails of such variables. The non-monotonicity is attributed to the detailed chemical structure of hydrogen-air flames where heat release occurs close to the unburnt reactants at near free stream temperatures.

Monday, November 25, 2013 10:30AM - 12:40PM –

Session H28 Waves III Spirit of Pittsburgh Ballroom B/C - Reza Alam, University of California, Berkeley

10:30AM H28.00001 A model for internal wave drift , FAN LIN, JAMES MUNROE, Memorial Univ of Newfoundland — We studied the motion of neuturally buoyant spheres induced by internal waves in a linearly stratified fluid with moderate Reynolds number (200-300). The characteristic scale of the sphere is much smaller than the wavescale ($D/\lambda < 0.05$) so we apply the Morison equation to model the motion of the spheres. In our 5-metre long wave tank, a mode-1 internal wave was generated by a wave generator to study the motion of the spheres. Experimental results show that similar to surface waves, there exists a wave induced drift of the sphere resulting from the phase lag between the motion of the sphere and the fluid. The magnitude and direction of the drift velocity u_d can be affected by many parameters, including the initial phase of the wave generator, depth of the sphere, and the frequency of the internal waves. An empirical formula for u_d will be introduced and will be compared to the theoretical results from a numerical simulation. For the vertical motion of the sphere, both the experiment and numerical simulation show that at low frequency of the internal waves ($\omega/N < 0.2$), a series of harmonics of ω appear in the vertical motion.

10:43AM H28.00002 Weakly nonlinear models for internal waves , SHENGQIAN CHEN, University of Wisconsin at Madison, ROBERTO CAMASSA, Department of Mathematics, University of North Carolina at Chapel Hill — In the class of weakly nonlinear models for internal waves, some systems are solvable by the Inverse Scattering Transform (IST). However, these models have the drawback of being ill-posed, or highly oscillatory wavetrains may develop in the solution such as for the Korteweg de Vries equation, thereby preventing standard numerical approaches from achieving the desired accuracy. In this talk, we propose a regularized version of the ill-posed two-layer Kaup, and the solitary wave solution for the new model is provided. The particular nature of the ill-posedness of Kaup's system proves to be rather challenging for designing numerical solution algorithms, a situation that is completely by-passed by the new regularized Kaup system. We provide numerical evidence showing that our regularization has little influence on the prediction offered by IST: the soliton content of initial data based on Kaup's system is left basically intact by its regularized counterpart, as tested by the numerical simulations of the new model.

10:56AM H28.00003 Particle dispersion induced by random internal waves[1] , OLIVER BUHLER, Courant Institute of Mathematical Sciences, New York University, NICOLAS GRISOUARD, Stanford University, MIRANDA HOLMES-CERFON, Courant Institute of Mathematical Sciences, New York University — This is a theoretical and numerical study of quasi-horizontal particle dispersion along stratification surfaces induced by random internal waves at small amplitude. The original motivation was small-scale particle dispersion in the deep ocean, but the theory is more general. The novelty is the realization that a small amount of wave dissipation can have a large impact on the dispersion process as measured by the Taylor diffusivity, for example. Basically, weak dissipation greatly strengthens the Taylor diffusivity, a fact that had been mostly overlooked in the literature so far. Here we present a combination of simple linear and nonlinear stochastic models as well as of fully nonlinear 3d simulations of the continuously stratified Boussinesq equations that explore this new situation. Particular attention is paid to the different power-law scalings of the Taylor diffusivity with wave amplitude that are obtained under different models for the wave dissipation, eg either due laminar viscous dissipation or due to nonlinear wave breaking.

[1]NSF-CMG 1024180, NSF-DMS 1312159

11:09AM H28.00004 Internal Wave Breaking From Parametric Subharmonic Instability , JAMES MUNROE, Memorial University of Newfoundland — Parametric subharmonic instability is an energy transfer mechanism between internal waves from large to small spatial scales. In this type of resonant triad interaction, a parent wave of higher frequency destabilizes leading to the growth of two daughter waves with lower frequencies. In a laboratory experiment, a full-depth wave generator forces a high frequency vertical mode-1 wave and parametric subharmonic instability generates large amplitude, high vertical wave number waves.

11:22AM H28.00005 Internal Waves Generated By A Horizontally Moving Source In A Thermocline - A WKB Approach , LAURA BRANDT, CECILY KEPPEL, JAMES ROTTMAN, Science Applications International Corporation, DAVID BROUTMAN, Computational Physics Inc — A new, computationally efficient method is described for calculating the internal wavefield generated by a localized source moving horizontally within an ocean thermocline. The new method involves Fourier-space ray-tracing, instead of a more traditional Green's function approach with eigenfunction expansion. In addition to computational efficiency, the new method provides physical insight into how the wavefield is generated. The Fourier-space ray-tracing method reproduces all of the terms in the Green's function solution (not just the eigenfunctions) and provides a physical explanation of the significance of an eigenfunction derivative term in that solution. For validation, this new method is compared with and used to analyze and explain the various transverse and divergent wave modes observed in previously published experimental tank data.

11:35AM H28.00006 Stability of internal wave beams to three-dimensional modulations[1] , T.R. AKYLAS, MIT, T. KATAOKA, Kobe University — The linear stability of uniform, plane internal wave beams with locally confined spatial profile, in a stratified fluid of constant buoyancy frequency, is discussed. The associated eigenvalue problem is solved asymptotically, assuming perturbations of long wavelength relative to the beam width. In this limit, instability is found only for oblique disturbances which vary in the along-beam and the horizontal transverse directions. Progressive beams, which transport energy in one direction and are directly relevant to internal tides, are unstable if the beam steepness exceeds a certain threshold value, whereas purely standing beams are unstable even at infinitesimal steepness. A distinguishing feature of this three-dimensional modulational instability is the generation of circulating horizontal mean flows at large distances from the vicinity of the beam.

[1]Supported by NSF and KTC.

11:48AM H28.00007 Interactions between capillary wave turbulence and hydrodynamics turbulence , MICHAEL BERHANU, LEONARDO GORDILLO, TIMOTHEE JAMIN, ERIC FALCON, MSC Universite Paris Diderot, CNRS, UMR 7057 Paris — We report experiments on capillary wave turbulence at the air-water interface. The field of wave elevation is measured using Diffusing Light Photography method. When wave turbulence regime is reached, we observe power-law spectra of wave elevation, both in frequency and in wave number, whose exponents are found in agreement with the predictions of capillary wave turbulence theory, although some hypotheses are not fulfilled. By the means of a laser sheet, we complete our observations by measuring in the same conditions in a vertical plane, the space-time deformation of the free surface using a Radon transform and the corresponding velocity field using 2D PIV algorithms. We aim to characterize vorticity generation by the waves and interaction between wave turbulence and hydrodynamics turbulence. These phenomena could indeed increase strongly the effective dissipation of non-linear propagating waves.

12:01PM H28.00008 On the unsteady gravity-capillary wave pattern found behind a slow moving localized pressure distribution , N. MASNADI, J.H. DUNCAN, University of Maryland — The non-linear response of a water surface to a slow-moving pressure distribution is studied experimentally using a vertically oriented carriage-mounted air-jet tube that is set to translate over the water surface in a long tank. The free surface deformation pattern is measured with a full-field refraction-based method that utilizes a vertically oriented digital movie camera (under the tank) and a random dot pattern (above the water surface). At towing speeds just below the minimum phase speed of gravity-capillary waves ($c_{min} \approx 23$ cm/s), an unsteady V-shaped pattern is formed behind the pressure source. Localized depressions are generated near the source and propagate in pairs along the two arms of the V-shaped pattern. These depressions are eventually shed from the tips of the pattern at a frequency of about 1 Hz. It is found that the shape and phase speeds of the first depressions shed in each run are quantitatively similar to the freely-propagating gravity-capillary lumps from potential flow calculations. In the experiments, the amplitudes of the depressions decrease by approximately 60 percent while travelling 12 wavelengths. The depressions shed later in each run behave in a less consistent manner, probably due to their interaction with neighboring depressions.

12:14PM H28.00009 3D Solitons of Capillary-Gravity and Flexural-Gravity Waves , REZA ALAM, University of California, Berkeley — In the context of nonlinear water wave theory an intriguing question has always been if fully-localized 3D wave structures, counterparts of 2D solitons, can exist. These structures are important because, if exist, they can transport mass, momentum and energy over long distances. For pure gravity waves this possibility is already ruled out, but- as we will discuss- few limiting cases of capillary-gravity and flexural-gravity wave equations admit such solutions in the form of dromions and lumps. Here we show that weakly nonlinear flexural-gravity wave packets, such as those propagating on the surface of ice-covered waters, admit three-dimensional fully localized solutions in the form of dromions. This study is motivated by observations of (relatively) large amplitude localized waves deep inside the ice-pack in polar waters. For capillary-gravity wave classical theory obtains dromions for shallow-water and strong surface tension (Bond number, Bo, greater than 1/3). Here we show that capillary-gravity dromions exist beyond this limit for a broad range of finite water depths as well as for sub-critical Bond numbers, i.e. for Bo $< 1/3$.

12:27PM H28.00010 Capillary Gravity Waves over an Obstruction - Forced Generalized KdV equation , JEONGWHAN CHOI, Korea University, S.I. WHANG, Ajou University, SHU-MING SUN, Virginia Tech — Capillary gravity surface waves of an ideal fluid flow over an obstruction is considered. When the Bond number is near the critical value 1/3, a forced generalized KdV equation of fifth order is derived. We study the equation analytically and numerically. Existence and stability of solutions are studied and new types of numerical solutions are found.

Monday, November 25, 2013 10:30AM - 12:01PM –

Session H30 Compressible Flows I: DNS 408 - Todd Oliver, University of Texas at Austin

10:30AM H30.00001 The convergence of DNS results to the LIA solution in canonical shock-turbulence interaction , JAIYOUNG RYU, DANIEL LIVESCU, Los Alamos National Laboratory — The interaction between isotropic turbulence and a normal shock wave is studied using Direct Numerical Simulations, with all flow scales (including the shock width) accurately solved. The simulation domain is open-ended, in a reference frame where the shock is stationary and turbulence is fed through the inlet. Realistic turbulence is generated in separate stationary isotropic simulations, with background velocity matching the shock speed, to avoid the use of the Taylor hypothesis. The shock Mach numbers range from 1.1 to 2.2 and the microscale Reynolds numbers range from 10 to 50. The vortical mode dominates upstream of the shock and the simulations cover the parameter space from linear inviscid, close to the Linear Interaction Analysis (LIA) limit, to regimes dominated by nonlinear and/or viscous effects. This comprehensive coverage of the parameter space shows, for the first time, that turbulence quantities from DNS converge to the LIA solutions as the shock width becomes thinner than the turbulence scales. In this regime, the shock Mach number becomes the dominant parameter, consistent to the LIA prediction.

10:43AM H30.00002 Reynolds Stress Anisotropy and Vortex Structures in Compressible Homogeneous Turbulent Shear Flow , VAIBHAV BHUTORIA, GREGORY BLAISDELL, MUKUL RAO, Purdue University — Direct numerical simulations of compressible homogeneous shear flow using natural initial conditions are performed for a range of gradient and turbulent Mach numbers. A pseudo-spectral Fourier collocation method is used to perform the simulations. Compressibility effects result in a reduced growth rate of turbulent kinetic energy, predominantly through a reduced production rate. This is found to be parameterized better by the gradient Mach number than by the turbulent Mach number. From the Reynolds stress anisotropy tensor it is found that the reduced production rate is due to lower energy in the velocity fluctuations in the direction of the mean velocity gradient. Lumley's tensor invariant map shows that more compressible flows tend towards the 1-component turbulence state. High speed and low speed streaks associated with corrugated vortex sheets are found in these simulations. The mechanism of formation of the corrugated vortex sheets is investigated.

10:56AM H30.00003 High-speed laminar-turbulent boundary layer transition induced by a discrete roughness element[1] , PRAHLADH IYER, KRISHNAN MAHESH, University of Minnesota — Direct numerical simulation (DNS) is used to study laminar to turbulent transition induced by a discrete hemispherical roughness element in a high-speed laminar boundary layer. The simulations are performed under conditions matching the experiments of Danehy *et al.* (*AIAA Paper 2009–394, 2009*) for free-stream Mach numbers of 3.37, 5.26 and 8.23. It is observed that the Mach 8.23 flow remains laminar downstream of the roughness, while the lower Mach numbers undergo transition. The Mach 3.37 flow undergoes transition closer to the bump when compared with Mach 5.26, in agreement with experimental observations. Transition is accompanied by an increase in C_f and C_h (Stanton number). Even for the case that did not undergo transition (Mach 8.23), streamwise vortices induced by the roughness cause a significant rise in C_f until $20D$ downstream. The mean van Driest transformed velocity and Reynolds stress for Mach 3.37 and 5.26 show good agreement with available data. A local Reynolds number based on the wall properties is seen to correlate with the onset of transition for the cases considered.

[1]Partially supported by NASA

11:09AM H30.00004 The role of compressions and expansions in stationary compressible isotropic turbulence , SHRIRAM JAGANNATHAN, DIEGO DONZIS, Texas A&M University — A characteristic of turbulence that is unique to compressible flows is the presence of compressing and expanding fluid regions that correspond to negative and positive values of dilatation respectively. While considerable attention has been given to studying the role of compressions, the effect of expansions has not been investigated in any detail. We employ a large database of Direct Numerical Simulation of stationary compressible isotropic turbulence at Taylor Reynolds numbers up to 450 and a range of Mach numbers ($M_t \approx 0.1 - 0.6$) to examine the impact of compressions and expansions on the statistics of thermodynamic variables. Our results indicate that expansions affect the flow thermodynamics more significantly than equally strong compressions. While at low M_t thermodynamic variables are less likely to be affected by compressions and expansions, they tend to be altered significantly by expansions at high M_t. Expansions are less likely to appear as compared to compressions, but tend to produce an increase in the correlation between density and temperature at high M_t, which, as will be shown, affects the pressure fluctuations. The differences in flow statistics in regions of intense fluctuations for low and high M_t will also be discussed.

11:22AM H30.00005 Predictive Inner-Outer Wall Model for Hypersonic Turbulent Boundary Layers , PINO MARTIN, CLARA HELM, University of Maryland — The inner-outer predictive wall model of Mathis et al. (JFM 2011) is modified for hypersonic turbulent boundary layers. The model is based on a modulation of the energized motions in the inner layer by large scale momentum fluctuations in the logarithmic layer. Using direct numerical simulation (DNS) data of Mach 3 and Mach 7 turbulent boundary layers it is shown that this modulating effect exists in compressible conditions and at low Reynolds number. The model is extended to include also spanwise and wall-normal velocity fluctuations and is generalized for compressible flow through Morkovin scaling. Temperature fluctuations are modeled using an appropriate Reynolds Analogy. Density fluctuations are calculated from the temperature fluctuations using an equation of state and a linear scaling with Mach number. DNS data are used to obtain the universal signal and parameters. The model is tested by using the universal signal to reproduce the flow conditions of Mach 3 and Mach 7 turbulent boundary layer DNS data and comparing turbulence statistics between the modeled flow and the DNS data. This work is supported by the Air Force Office of Scientific Research under grant AF/9550-10-1-0164.

11:35AM H30.00006 A Semi-Implicit, Fourier-Galerkin/B-Spline Collocation Approach for DNS of Compressible, Reacting, Wall-Bounded Flow , TODD OLIVER, RHYS ULERICH, VICTOR TOPALIAN, NICK MALAYA, ROBERT MOSER, The University of Texas at Austin — A discretization of the Navier-Stokes equations appropriate for efficient DNS of compressible, reacting, wall-bounded flows is developed and applied. The spatial discretization uses a Fourier-Galerkin/B-spline collocation approach. Because of the algebraic complexity of the constitutive models involved, a flux-based approach is used where the viscous terms are evaluated using repeated application of the first derivative operator. In such an approach, a filter is required to achieve appropriate dissipation at high wavenumbers. We formulate a new filter source operator based on the viscous operator. Temporal discretization is achieved using the SMR91 hybrid implicit/explicit scheme. The linear implicit operator is chosen to eliminate wall-normal acoustics from the CFL constraint while also decoupling the species equations from the remaining flow equations, which minimizes the cost of the required linear algebra. Results will be shown for a mildly supersonic, multispecies boundary layer case inspired by the flow over the ablating surface of a space capsule entering Earth's atmosphere. This work is supported by the Department of Energy [National Nuclear Security Administration] under Award Number [DE-FC52-08NA28615].

11:48AM H30.00007 Direct Numerical Simulation of a Compressible Reacting Boundary Layer using a Temporal Slow Growth Homogenization , VICTOR TOPALIAN, TODD OLIVER, RHYS ULERICH, ROBERT MOSER, The University of Texas at Austin — A DNS of a compressible, reacting boundary layer flow at $\mathrm{Re}_\theta \approx 430$ was performed using a temporal slow-growth homogenization, for a multispecies flow model of air at supersonic regime. The overall scenario parameters are related to those of the flow over an ablating surface of a space capsule upon Earth's atmospheric re-entry. The simulation algorithm features Fourier spatial discretization in the streamwise and spanwise directions, B-splines in the wall normal direction, and is marched semi-implicitly in time using the SMR91 scheme. Flow statistics will be presented for relevant flow quantities, in particular those related with RANS modeling. Since analogous slow growth computations can be performed using RANS to predict the flow mean profiles, the use of data gathered from this type of simulation as a vehicle for the calibration and uncertainty quantification of RANS models will be discussed. This work is supported by the Department of Energy [National Nuclear Security Administration] under Award Number [DE-FC52-08NA28615].

Monday, November 25, 2013 10:30AM - 12:14PM –

Session H31 Porous Media Flows VI: Imbibition and Injection 402 - Carlos H. Hidrovo, University of Texas at Austin

10:30AM H31.00001 Droplet Impact and Penetration on a Series of Capillary Tubes , SAMAN HOSSEINI, ALIREZA DALILI, NASSER ASHGRIZ, SANJEEV CHANDRA, University of Toronto — A series of experiments were carried out in which a single droplet of water was deposited onto a substrate having a series of closely spaced parallel-holes to represent a simple porous media. At the center of the width of the 0.6"x0.5"x0.3" poly-carbonate substrate seven through-holes each with a diameter of 300 μm and distance of 300 μm from one another were drilled in a straight line. Droplets with diameters of 3.2 and 2.0 mm were released from heights of 1, 3 and 5 cm. Using a high-speed camera the impact, spreading and capillary penetration of the droplets into the holes were videotaped. Two different penetration regimes were observed based on the impact velocity. At low droplet impact velocities, the penetration was mainly due to capillary forces, while at higher impact velocities the penetration occurred at two stages. The first stage was inertia driven, while the second stage was capillary driven penetration. The threshold velocity for liquid penetration into the holes was formulated. Inertia forces were used to describe the linear portion of penetration and the Lucas-Washburn equation was used to characterize the non-linear (capillary) part of penetration. The distance of penetration as a function of time was worked out using this equation. Droplet oscillation on the top of the parallel holes was observed as well. It was evident that the area of the penetration inside the holes played a major role in the kinetic energy dissipation and the damping of the oscillation.

10:43AM H31.00002 Design of Capillary Flows with Spatially Graded Porous Films , YOUNG SOO JOUNG, BRUNO MICHEL FIGLIUZZI, CULLEN BUIE, Massachusetts Institute of Technology — We have developed a new capillary tube model, consisting of multi-layered capillary tubes oriented in the direction of flow, to predict capillary speeds on spatially graded porous films. Capillary flows through thin porous media have been widely utilized for small size liquid transport systems. However, for most media it is challenging to realize arbitrary shapes and spatially functionalized micro-structures with variable flow properties. Therefore, conventional media can only be used for capillary flows obeying Washburn's equation and the modifications thereof. Given this background, we recently developed a method called breakdown anodization (BDA) to produce highly wetting porous films. The resulting surfaces show nearly zero contact angles and fast water spreading speed. Furthermore, capillary pressure and spreading diffusivity can be expressed as functions of capillary height when customized electric fields are used in BDA. From the capillary tube model, we derived a general capillary flow equation of motion in terms of capillary pressure and spreading diffusivity. The theoretical model shows good agreement with experimental capillary flows. The study will provide novel design methodologies for paper-based microfluidic devices.

10:56AM H31.00003 Phase-field modeling of two-phase flow in porous media with partial wetting , LUIS CUETO-FELGUEROSO, RUBEN JUANES, Massachusetts Institute of Technology — Current models of multiphase flow in porous media implicitly assume complete wetting, and are unable to describe non-spreading systems. This limitation has a direct impact on the ability of current theories to predict complex non-equilibrium processes in porous media, such as flow instabilities or rate-dependent displacement patterns. Here we present a continuum model of two-phase flow in porous media that can describe partially wetting systems. The model is derived within the framework of phase-field modeling. We study unstable two-dimensional flow due to viscous fingering (the instability that ensues when a less viscous fluid displaces a more viscous one in a porous medium). The displacement pattern is characterized by branching structures, with an intrinsic length scale that depends on the fluid properties, essentially viscosity and surface tension between the fluids, as well as the structure of the porous space, the wetting properties of the system, and the injection rate. Using our macroscopic model, we discuss the scaling properties of the intrinsic finger length scale.

11:09AM H31.00004 Saffman-Taylor fingering with lateral injection with applications to imbibition coarsening dynamics , BERTRAND LAGREE, TOTAL SA/Institut Jean le Rond d'Alembert, STEPHANE ZALESKI, Institut Jean le Rond d'Alembert/CNRS - Universite Pierre et Marie Curie (UMR 7190), IGOR BONDINO, TOTAL SA, CHRISTOPHE JOSSERAND, Institut Jean le Rond d'Alembert/CNRS - Universite Pierre et Marie Curie (UMR 7190), STEPHANE POPINET, Institut Jean le Rond d'Alembert/CNRS - Universite Pierre et Marie Curie (UMR 7190)/NIWA — We report 2D simulations of Saffman-Taylor fingering motivated by the analysis of experiments on the imbibition of porous media in square slab geometries. We use a Volume-of-Fluid (VOF) method to model a two-phase Darcy flow with a sharp interface between the two fluids. The Gerris code which allows efficient parallel computations with quad-tree mesh refinement is used. It is tested for accuracy and precision using several levels of refinement and comparing to reference simulations in the literature. A fingering pattern is observed after lateral injection of a less viscous fluid into a region filled with a more viscous one. Large fractal-like clusters are observed allowing the measurements of several scaling exponents which are compared to the known Diffusion-Limited-Aggregation (DLA) and Saffman-Taylor scalings. An interesting effect is the transition from a transient cylindrical DLA pattern to a small number then a single Saffman Taylor finger.

11:22AM H31.00005 Mechanics of fluid injection into a soft granular material , CHRISTOPHER MACMINN, ERIC DUFRESNE, JOHN WETTLAUFER, Yale University — Motivated by a range of problems in geophysics and biology where fluid injection drives the mechanical deformation of a porous solid, we perform laboratory experiments in a model system. We inject fluid into a packing of soft particles and measure the dynamic, flow-driven deformation of the packing at high spatial resolution. We show that the mean deformation and relaxation of the packing, as well as the buildup and dissipation of pressure, can be described by continuum poroelastic theory. We also find, in contrast, that the granular microstructure leads to the spontaneous emergence of heterogeneous mesoscale features such as shear bands that are absent from the continuum theory. We discuss the implications of these results.

11:35AM H31.00006 The usage of differential method in determining the multiphase flow transport parameters in porous media , BOJAN MARKICEVIC, Pall Corp — The imbibition of wetting liquid by a porous medium starts as a single-phase flow which later transforms into a multiphase flow pattern as wetting liquid progresses into the medium. These capillary flows can be solved using the dynamic capillary network models, where the capillary pressure is calculated at the liquid free interface and progression of the flow front is found from fully resolved velocity and pressure profiles within the wetted domain. From known flow quantities, both phase permeability and capillary pressure are determined as a function of a spatial position in the flow geometry. The phase content (saturation) is also calculated from the numerical solution, and after correlations, the phase permeability and capillary pressure as a function of saturation are found. Two independent checks of this differential method are carried out: the first one being the invariance of the single-phase permeability. For region next to the fluid inlet, it is shown that the pressure gradient and the flow rate are always linearly dependent irrespective of the flow front position downstream. Secondly, the phase permeability and capillary pressure saturation functions should not change throughout the spread, irrespective of the time in which they are measured, but rather they should follow the same dependence on the saturation. The numerical results corroborate this assumption, where the invariant permeability and capillary pressure laws are predicted throughout the imbibation duration. Finally, additional properties of the porous medium can be determined including a minimum saturation of the percolation cluster within porous medium.

11:48AM H31.00007 Pore-scale modeling of Capillary Penetration of Wetting Liquid into 3D Fibrous Media: A Critical Examination of Equivalent Capillary Concept , NIKHIL KUMAR PALAKURTHI, URMILA GHIA, University of Cincinnati, KEN COMER, Procter and Gamble — Capillary penetration of liquid through fibrous porous media is important in many applications such as printing, drug delivery patches, sanitary wipes, and performance fabrics. Historically, capillary transport (with a distinct liquid propagating front) in porous media is modeled using capillary-bundle theory. However, it is not clear if the capillary model (Washburn equation) describes the fluid transport in porous media accurately, as it assumes uniformity of pore sizes in the porous medium. The present work investigates the limitations of the applicability of the capillary model by studying liquid penetration through virtual fibrous media with uniform and non-uniform pore-sizes. For the non-uniform-pore fibrous medium, the effective capillary radius of the fibrous medium was estimated from the pore-size distribution curve. Liquid penetration into the 3D virtual fibrous medium at micro-scale was simulated using OpenFOAM, and the numerical results were compared with the Washburn-equation capillary-model predictions. Preliminary results show that the Washburn equation over-predicts the height rise in the early stages (purely inertial and visco-inertial stages) of capillary transport.

12:01PM H31.00008 Optimization of Micropillar Arrays for Heat Pipe Applications[1] , RENEE HALE, CARLOS HIDROVO, The University of Texas at Austin — Demand is rising for improved thermal management solutions in areas such as electronics cooling. Heat pipes are an attractive technology, but their cooling capacity is limited by the maximum flow rate that their internal wicking structure can sustain. This capillary limit depends upon the interplay between the permeability of the internal wicking structure and the capillary forces produced by the wick pores. Micropillar arrays have recently received attention as potential wicking materials, and this project seeks to design, manufacture, test, and optimize micropillar arrays for heat pipe applications. The novelty of this work resides in the exploration of rectangular pillar arrangements where the pillar spacing is not identical in both directions. This work utilizes analytical and numerical models of fluid flow to determine array permeability. The capillary pressure is predicted by surface energy minimization techniques. Pillar dimensions are then optimized to obtain the maximum fluid flow rate through the wick. To test the wicks, a thermo-hydraulic characterization setup directly measures the mass flow rate of a working fluid through a wicking material as a function of applied heat load. The results give a clear indication of the heat capacity of each wick and provide a valuable connection between experimental results and model predictions for the fluid velocity. This work tests a range of micropillar array geometries and reports on their suitability as wicking structures for heat pipe applications.

[1]Support provided by NSF Grant 1134104

Monday, November 25, 2013 10:30AM - 12:27PM –

Session H32 Viscous Flows II: Flows in Viscous Fluids 403 - David Saintillan, University of Illinois at Urbana-Champaign

10:30AM H32.00001 Motion of a Bellows and a Free Surface in a Closed Vibrated Liquid-Filled Container , J.R. TORCZYNSKI, L.A. ROMERO, T.J. O'HERN, Sandia National Laboratories — The coupled motion of a bellows and an idealized free surface in a closed container that is filled with an incompressible viscous liquid and that is vibrated vertically is investigated computationally and theoretically. The bellows and the free surface exhibit rectified motion in the sense that their displacements from their equilibrium positions averaged over a cycle are nonzero. Two types of rectification that arise from two sources of nonlinearity are identified. "Geometric rectification" results from the time variation of the bellows/free-surface geometry. "Advective rectification" results from the advection term in the Navier-Stokes equations. An approximate theory based on these ideas agrees well with direct numerical simulations over a broad range of frequencies from well below to well above resonance. 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.

10:43AM H32.00002 On the inertial motions of liquid-filled rigid bodies , GIUSY MAZZONE, GIOVANNI GALDI, PAOLO ZUNINO, University of Pittsburgh — We consider a rigid body with a cavity completely filled by a viscous liquid and study the inertial motions of the system liquid-filled rigid body $\mathcal{S}$. The equations governing the motion of this coupled system are given by the Navier-Stokes equations and the equations of the balance of the total angular momentum of $\mathcal{S}$ in absence of external forces and torques. Given any initial motion to the coupled system, characterized by an initial relative velocity of the fluid and an initial total angular momentum, we give a complete description of the behavior that the system liquid-filled rigid body will show at large times. From both analytical and numerical viewpoints, we are able to prove a longstanding conjecture stated by Zhukovskii, namely that $\mathcal{S}$ will eventually reach a steady state which is a rigid body permanent rotation. In other words, the liquid goes to rest with respect to the rigid body and the coupled system will rotate as a whole rigid body, with a constant angular velocity that is directed along one of the principal axes of inertia of the system.

10:56AM H32.00003 Trajectory and flow properties for a rod spinning in a viscous fluid: An asymptotic solution with a no-slip plane , LONGHUA ZHAO, Case Western Reserve University, ROBERTO CAMASSA, University of North Carolina at Chapel Hill, TERRY JO LEITERMAN, St. Norbert College, RICHARD MCLAUGHLIN, LEANDRA VICCI, University of North Carolina at Chapel Hill — Utilizing singularity theory and the slender body theory, asymptotic solutions are constructed for a slender body sweeping out a double cone or single cone in free space in the low Reynolds number regime. The asymptotic solutions are compared quantitatively with the exact solutions for a prolate spheroid performing similar motion. With a set of singularities, Blakelet, an asymptotic solution is developed for a slender cylinder attached to a no-slip plane and sweeping out an upright cone. The no-slip boundary condition is satisfied exactly. Trajectory and flow properties are examined with special attention paid to the case study on slenderness between the exact and asymptotic free space solutions. For flow with no-slip plane, the theoretical prediction is compared with the experimental data, which shows good agreement. Far field asymptotic analysis is presented for the asymptotic velocity when the slender body precesses cones in free space and with no-slip plane. When the cone is tilted, the asymptotic solution is constructed in the lab frame, and the experimental data are reported compared with the theoretical prediction. This study is of direct use to nano-scale actuated fluidics where similar epicyclical behavior has been observed.

11:09AM H32.00004 Three-dimensional Developing Flow in a Long Serpentine Channel , SURYA P. VANKA, Department of Mechanical Science and Engineering, University of Illinois at Urbana-Champaign — Serpentine wavy channels are often used as fin passages in compact heat exchangers because of their increased heat transfer performance. However, their benefit is seen only in the unsteady flow regime, and in the turbulent regime. In this work, we study the three-dimensional developing flow in a wavy passage using a finite volume fractional-step Navier-Stokes solver. The geometry consists of a straight approach section, several (8) waves and a stratight section at the end. A curvilinear grid is used to represent the flow domain. Both the steady and unsteady flow regimes are computed by systematically increasing the Reynolds numbers. The effects of the wave amplitude and the wave length are also studied. The structure of the developing flow is presented for different parameter selections. The spanwise structure of the flow on the curved surfaces, and the formation of the recirculating regions in the troughs are presented. Pressure drop characteristics in the developing region are compared with those in the fully-developed periodic region.

11:22AM H32.00005 The sedimentation of flexible filaments: Shapes, trajectories, and clouds , SAVERIO SPAGNOLIE, LEI LI, University of Wisconsin-Madison, HARISHANKAR MANIKANTAN, DAVID SAINTILLAN, University of Illinois at Urbana-Champaign — The dynamics of a flexible filament sedimenting in a viscous fluid (Stokes flow) are investigated. Compared to the well-studied case of sedimenting rigid rods, the introduction of filament compliance is shown to cause a significant alteration in the long-time sedimentation orientation and filament geometry. A model is developed by balancing viscous, elastic, and gravitational forces in a slender-body theory, and the filament dynamics are characterized by a dimensionless elasto-gravitation number. In the weakly flexible regime, a multiple-scale asymptotic expansion is used to obtain expressions for filament translations, rotations, and shapes, which match excellently with full numerical simulations. Furthermore, trajectories of sedimenting flexible filaments, unlike their rigid counterparts, are restricted to a cloud whose envelope is determined by the elasto-gravitation number.

11:35AM H32.00006 The sedimentation of flexible filaments: A buckling instability , HARISHANKAR MANIKANTAN, DAVID SAINTILLAN, University of Illinois at Urbana-Champaign, LEI LI, SAVERIO SPAGNOLIE, University of Wisconsin-Madison — An elastic filament sedimenting in a viscous fluid can lead to complex deformations and dynamics due to the non-trivial interplay between gravity, viscous stresses and its internal elastic forces. One such dramatic case is the buckling of a flexible filament placed with its long axis parallel to gravity. Using slender-body theory for low-Reynolds number flows, we first show that a non-uniform tension is induced in the filament due primarily to a non-uniform shape and secondarily to non-local hydrodynamic interations. This tension acts to compress the filament in its leading half and can lead to a buckling instability in the highly flexible regime, which we characterize via a dimensionless elasto-gravitation number. We derive a dispersion relation that clearly illustrates this competing effect between tension and elastic rigidity, and also suggests that the instability travels as waves in the direction opposite gravity. We then turn to numerical simulations to verify this, and see that waves grow and travel as predicted. We also look at linear eigenmodes of the governing equation, and the mode shapes so obtained agree well with those observed in simulations.

11:48AM H32.00007 Quantitative Viscosity Field Measurement during Viscous Fingering by Imaging Fluorescence from a Viscosity-Sensitive Molecular Probe , BRADLEY DICE, MICHAEL RAWAT, SIMONE STEWART, PATRICK BUNTON, Department of Physics, William Jewell College, Liberty, MO, FABIAN BRAU, ANNE DE WIT, Nonlinear Physical Chemistry Unit, Université Libre de Bruxelles, Bruxelles, Belgium, JOHN POJMAN, Department of Chemistry, Louisiana State University, Baton Rouge, LA — The two-dimensional spatio-temporal distribution of the viscosity field has been measured quantitatively during radial displacements of pure glycerol and a miscible solution of glycerol and water inside a horizontal Hele-Shaw cell. Ultraviolet-excited fluorescence from the viscosity-sensitive molecular probe Auramine O was imaged in situ during the displacement. Fluorescence intensity as a function of viscosity was calibrated using known values of viscosity for glycerol-water solutions from the literature. From this calibration, the two-dimensional spatio-temporal map of fluorescence allowed for reconstruction of the evolution of the viscosity field during either the stable displacement or the viscous fingering process. For the stable case the viscosity profile was compared to the known analytical solution. This technique should prove widely applicable for in situ viscosity measurements during flow instabilities subject to appropriate choice of molecular probe.

12:01PM H32.00008 Low Reynolds number hydrodynamics of microstructured optical fiber fabrication[1] , PETER BUCHAK, DARREN CROWDY, Imperial College London, YVONNE STOKES, University of Adelaide — Microstructured optical fibers (MOF's) derive novel optical capabilities from having large numbers of wavelength-scale channels. MOF's are fabricated by the capillary drawing of a molten glass preform at low Reynolds number, during which the cross section deforms under surface tension, with the result that the configuration of the channels in the fiber may differ from the perform. This unintended deformation is inadequately understood and is difficult to investigate experimentally. In this talk, we describe methods we have developed to model slender viscous fibers with multiply connected cross section, which make possible theoretical investigation of the deformation, with the aim of determining the preform configuration required to produce a fiber with a desired arrangement of channels.

[1]Support from the Leverhulme Trust is gratefully acknowledged.

12:14PM H32.00009 Microstructured optical fibres: how do physical parameters influence the final geometry?[1] , YVONNE STOKES, The University of Adelaide, PETER BUCHAK, DARREN CROWDY, Imperial College London — Drawing of a microstructured optical fibre can be described by a 1D differential equation for the cross-sectional area as a function of axial position and a 2D classical Stokes-flow model for the evolution of the cross-sectional geometry as it moves along the fibre axis. These two models are coupled through the total length of the cross-sectional boundary. Physical parameters, including the initial preform geometry, the draw ratio and material properties, enter the model in non-trivial ways and affect the final fibre geometry. In this talk we will examine the coupled 1D and 2D models of fibre drawing to gain understanding of the influence of physical parameters on the final geometry, and also look at the existence and uniqueness of solutions. We aim to determine what fibre geometries can or cannot be obtained from a given initial preform.

[1]Support from the Australian Research Council (DP130101541) and the Leverhulme Trust is gratefully acknowledged.

Monday, November 25, 2013 10:30AM - 12:27PM –

Session H33 Drops IX: Evaporating Sessile Drops 404 - Pirouz Kavehpour, University of California, Los Angeles

10:30AM H33.00001 Dynamics of evaporating sessile droplets[1] , PEDRO SÁENZ, PRASHANT VALLURI, KHELLIL SEFIANE, University of Edinburgh, GEORGE KARAPETSAS, University of Thessaly, JUNGHO KIM, University of Maryland, College Park, OMAR MATAR, Imperial College London — A sessile droplet laying on a horizontal substrate evaporates into its surrounding gas. The dynamics of this physical system are investigated by means of 3D Direct Numerical Simulations and experiments. A non-isothermal two-phase model is employed to compute the spatio-temporal evolution of the system under consideration. Transient species transport in the gas phase is also accounted for via the general advection-diffusion governing equation. The interface mass transfer is computed considering that the vapour diffusion is the rate-limiting mechanism. On this premise, it is assumed that the liquid and the gas maintain thermodynamic quasi-equilibrium at the interface. The same system is also experimentally investigated by simultaneously recording the droplet evaporation in a controlled environment with a CCD camera (side) and an IR camera (top). Comparisons between numerical and experimental data are presented along with a discussion of the role played by other singularities of the system, namely the triple line, the effect of thermocapillarity, etc.

[1]Fundación Caja Madrid, EPSRC DTA & ThermaPOWER (EU IRSES-PIRSES GA-2011-294905)

10:43AM H33.00002 ABSTRACT WITHDRAWN –

10:56AM H33.00003 On the lifetimes of evaporating droplets , STEPHEN WILSON, JUTTA STAUBER, BRIAN DUFFY, University of Strathclyde, Glasgow, UK, KHELLIL SEFIANE, University of Edinburgh, Edinburgh, UK — The evaporation of a fluid droplet on a solid substrate is a practically important problem which has been the subject of considerable research in recent years, much of it motivated by a range of technological applications, such as the application of pesticides to plants, DNA microarray analysis, inkjet printing, micro-fabrication, and spray cooling. In particular, the lifetime of a fluid droplet is not only of fundamental scientific interest, but is also important in a number of technological applications, such as inkjet printing and spray cooling applications (in which shorter droplet lifetimes are often needed) and the application of pesticides to plants (in which longer droplet lifetimes are often needed). In this talk we will analyse the lifetimes of fluid droplets evaporating in a variety of modes and, in particular, show that the widely believed folklore that the lifetime of a droplet is always longer than that of an identical droplet evaporating in the constant radius (i.e. pinned contact line) mode and shorter than that of an identical droplet evaporating in the constant angle mode is not, in general, true.

11:09AM H33.00004 Contact line and bulk velocities in evaporating micron-scale droplets , YI FAN, KENNETH S. BREUER, Brown University — The famous "coffee stain" phenomena is well known during the evaporation of a liquid droplet seeded with colloidal particles. However, the different phases of evaporation on a hydrophobic surface have not yet been fully explored and explained. In this experiment, evaporating micro-drops (diameter $\sim$100 μm) are seeded with 300 nm red fluorescent particles and observed from below using epifluorescent microscopy. We observed four phases: (i) steady evaporation with the contact line retreating at a constant speed of $\sim$2 μm/s; (ii) stagnation of the contact line for several seconds; (iii) a sharp transition leading to fast evaporation with a rapidly retreating contact line, and (iv) final dry-out of the film leaving the particles immobilized on the substrate. These four phases of motion, as well as the pattern of the deposited nano-particles are strong functions of both the colloidal concentration and the static contact angle. Statistical Particle Tracking Velocimetry is used to quantify the velocity fields inside the micro-drop during the evaporation history.

11:22AM H33.00005 Volumetric thermal measurements using thermo-liquid crystal (TLC) micro-particles in evaporating drops , RODRIGO SEGURA, ALVARO GOMEZ MARIN, CHRISTIAN KAEHLER, Institute for Fluid Mechanics and Aerodynamics, Bundeswehr University Munich — Freely evaporating sessile droplets develop weak temperature gradients that can generate Marangoni flows at the drop's surface. Quantitative temperature measurements of small gradients at such scales are very difficult. In this work, a method to track the temperature of individual thermo-liquid crystal (TLC) particles is employed to extract the temperature field inside an evaporating droplet. TLC thermography has been investigated for several years but the low quality of individual TLC particles, as well as the methods used to extract temperature from their color appearance, has prevented the development of a reliable approach to track their temperature individually. In order to overcome these challenges, an emulsion of stable non-encapsulated TLC micro particles with a narrower size distribution than that of commercial encapsulated TLC solutions was used along with a multi-variable calibration approach, as opposed to the direct hue-temperature relationship usually implemented (Segura et al, Microfluid Nanofluid, 2012). In addition, an optimized color space was implemented as well as circular polarization filtering to remove background noise and improve signal-to-noise ratio. Using this technique, a 3D temperature-velocity field within a droplet could be simultaneously resolved.

11:35AM H33.00006 Marangoni or not Marangoni? Thermal Marangoni flow measurements in evaporating drops , ALVARO GOMEZ MARIN, ROBERT LIEPELT, MASSIMILIANO ROSSI, CHRISTIAN KAEHLER, Bundeswehr University Munich — Sessile evaporating droplets fascinate for the rich and complex behavior that hides behind their apparent simplicity. Although the basic physics of the coffee-stain formation can be explained assuming thermal equilibrium (Deegan, 1997), thermal effects play an important role in the flow patterns within the droplet and in the deposits left on the substrate. Understanding such flows would give a chance to add a higher degree of control in these not-so-simple systems. For example, several studies have recently suggested that such thermal Marangoni flows can be strong enough to neutralize the coffee-stain effect. Experimental work in this sense has been scarce due to the difficulty of tracking particles at the surface of the droplet, where the flow is originated. In this study we perform fully three-dimensional and time resolved particle tracking measurements of particles suspended in sessile drops of liquids on substrates with different thermal conductivity ratios. The results are compared with some of the theoretical models and simulations available in the literature. Our final aim is to precisely quantify how important is the thermal Marangoni flow in an evaporating drop and if it can be used for practical applications.

11:48AM H33.00007 Approximate analytical descriptions of the stationary single-vortex Marangoni convection inside an evaporating sessile droplet of capillary size , LEV BARASH, Landau Institute for Theoretical Physics — Three versions of an approximate analytical description of the stationary single vortex Marangoni convection in an axially symmetrical sessile drop of capillary size are studied for arbitrary contact angle and compared with the results of numerical simulations. The first approach is heuristic extension of the well-known lubrication approximation. Two other new descriptions are developed for arbitrary contact angle and named nr- and rz-description. They are free from most of restrictive assumptions of the lubrication approach. For droplets with large contact angles they result in better accuracy compared to the heuristic extension of the lubrication approach, which still gives reasonable results within the accuracy 10-30 per cent. For droplets with small contact angles all three analytical descriptions well agree with the numerical data.

12:01PM H33.00008 The effect of vapor diffusion on the evaporation of a sessile droplet on a heated substrate , MAHNPRIT JUTLEY, VLADIMIR AJAEV, Southern Methodist University — The study of the physical behavior of sessile droplets on heated substrates is important for many applications, such as the coating of a solid substrate with another material or the spray cooling of electronics. In order to simulate the height evolution of the droplet and its effect on the temperature distribution in the substrate, a model that incorporates the effects of surface tension, gravity, evaporation, thermocapillarity, and disjoining pressure must be used. Due to the physical characteristics of a thin sessile droplet, a lubrication-type model that includes the aforementioned effects can be used. By solving the heat equation in the substrate, the lubrication-type equations in the droplet, and the quasi-steady diffusion equation in the gas phase, we simulate the effect of vapor diffusion on the evaporation of a sessile droplet and its coupling to the pattern of heating in the substrate. By using high-order numerical techniques for solving governing partial differential equations, the height evolution of the droplet, heat distribution in the substrate, and vapor diffusion over time are calculated. Connection of our predictions to recent experimental studies is discussed.

12:14PM H33.00009 How gravity influences hydrothermal waves in alcohol sessile droplets[1] , FLORIAN CARLE, DAVID BRUTIN, Aix Marseille University - IUSTI — This study, performed under several gravity levels, focuses on the formation and behaviour of hydrothermal waves (HTWs) that spontaneously develop on droplets surfaces when surface tension gradient are strong enough as a response to temperature gradients. HTWs have been found to form concentric torus around the apex rotating in the same direction from hot to cold area creating a shear phenomenon between the two torus where instability develops and get dragged in the flow. This leads to a detachment of the thermal plume, well visible on the top infrared visualisation. HTWs develop in a large number in methanol, half as much in ethanol and at this day, no instability have been experimentally observed in propanol. These behaviours are evidenced with the effective Marangoni number; its high value for the methanol indicates an intense turbulent flow. Gravity levels influence the atmospheric convective transport contribution to the droplets evaporation inducing diffusive evaporation under microgravity conditions and diffusive and convective evaporation under Moon, Mars and Earth gravity level; convection being directly linked to buoyancy. Evaporation dynamics is therefore modified and the temperature gradient between the contact line and the apex can be more or less important.

[1]The authors acknowledge the financial support of the "Centre National d'Études Spatiales" and also for the parabolic flights conducted at Bordeaux Merignac, France.

Monday, November 25, 2013 10:30AM - 12:01PM –

Session H34 Drops X: Splashing on Heated Surfaces 405 - Vladimir Ajaev, Southern Methodist University

10:30AM H34.00001 Splash transition of droplets impacting on heated surfaces , HENDRIK J.J. STAAT, TUAN TRAN, BART GEERDINK, CHAO SUN, DETLEF LOHSE, University of Twente — For large enough velocities, droplets impacting on a dry solid surface make a splash. How does the surface temperature affect the transition towards the splashing regime? We answer this questions by high-speed interferometric imaging for millimeter-sized droplets. We find that for moderate surface heating when the droplet still touches the surface, the velocity threshold towards splashing increases with increasing surface temperature. In contrast, for strong surface heating when the droplet is in the Leidenfrost regime and does not touch the superheated surface due to the formation of a vapor layer, the velocity threshold towards splashing is much lower. We theoretically explain both findings within a pressure balance model.

10:43AM H34.00002 Microdroplet impact on superheated surfaces: Vapor triggers splashing , TUAN TRAN, YOSHIYUKI TAGAWA, Physics of Fluids, University of Twente, YANBO XIE, MESA+ institute for nanotechnology, University of Twente, CHAO SUN, DETLEF LOHSE, Physics of Fluids, University of Twente — In many engineering and technological applications that involve impact of microdroplets on a superheated solid surface, the small size of the droplets and the vaporization of the liquid as the droplets approach the surface pose a challenge to visualize and understand the splash mechanism. In particular, the spontaneously generated vapor contributes to destabilize the spreading stage of the liquid and potentially influences the onset of splash. In this study, we experimentally determine the dependence of the splash transition on the surface temperature. We also provide a model that incorporates the liquid-vapor phase transition to explain this experimentally-observed transition.

10:56AM H34.00003 The Effect of Disturbances and Surrounding Air on the Droplet Impact Phenomena , ANDREW WORK, YONGSHENG LIAN, University of Louisville, MARK SUSSMAN, University of Florida — Supercooled Large Droplets (SLDs) represent an icing hazard in a number of areas, most obviously in aviation. SLDs pose a hazard above smaller supercooled droplets because they don't freeze completely on impact, and can spread or splash. Experiments have demonstrated that surrounding air plays an important role in the droplet impact phenomena: a low ambient pressure can suppress the droplet splashing. However, the effect of surrounding air on the droplet impact has not been adequately addressed. Numerical simulations are conducted to systematically investigate the interplay between the droplet and the surrounding air in the droplet splashing regime. Disturbances originating from the experimental droplet generator are also studied in the simulation. We investigate whether these disturbances are responsible for the fingering observed in experimentation. We compare the results of several perturbations on the droplet, as well as the effect of surface roughness. Simulations are conducted using the Moment of Fluid numerical method, and the grid features adaptive mesh refinement.

11:09AM H34.00004 How drops bounce and dance on ice: the role of sublimating surfaces[1] , CARLO ANTONINI, ETH Zurich, ILARIA BERNAGOZZI, University of Bergamo, STEFAN JUNG, DIMOS POULIKAKOS, ETH Zurich, MARCO MARENGO, University of Bergamo, LABORATORY OF THERMODYNAMICS IN EMERGING TECHNOLOGIES TEAM, THEMAL PHYSICS LABORATORY TEAM — Drop rebound is a spectacular event that appears after impact on superhydrophobic surfaces, due to low drop-solid substrate adhesion, and on hot substrates in Leidenfrost conditions, thanks to a vapor layer forming at the liquid-substrate interface, caused by drop evaporation. However, at temperatures below water freezing temperature, i.e. 0C, even superhydrophobicity can get lost. In the present work, we demonstrate that drop rebound can also be originated by another physical phenomenon, i.e. the solid substrate sublimation, at temperatures as low as -79C. To prove this mechanism, drop impact experiments were conducted on solid carbon dioxide, commonly known as dry ice. Drop dynamics and rebound were analyzed, together with the cases of drop impacting on a superhydrophobic surface and on a hot plate, to show how three different physical mechanisms, which apparently share nothing in common, i.e. superhydrophobicity, evaporation and sublimation, can all lead to drop rebound, in an extremely wide temperature range, from 300C down to even below -79C. Additional glycol drop impact tests proved the independence of the observed phenomena from the chosen liquid. Finally, the formation and visualization of an air vortex ring around an impacting drop is also reported.

[1]The authors acknowledge funding from Regione Lombardia and European Community (Marie Curie Fellowship)

11:22AM H34.00005 Quantitative visualization of droplet hot-surface interaction , NEJDET ERKAN, KOJI OKAMOTO, The University of Tokyo — Up to this date liquid droplet impingement phenomenon onto hot surfaces has drawn massive attention from a broad spectrum of research fields, since its hydrodynamic and thermodynamic characteristics has profound importance for various industrial applications Although tremendous experimental and computational work exist in the literature, thermal-hydraulic mechanism of droplet impingement boiling on hot surfaces received several contradictory approaches due to the parametric sensitivity of the problem. To understand and to predict the physical mechanism, an experimental database including large amount of spatio-temporal data, which is formed by the tests performed under well-controlled BCs and high sensitive devices, is still a necessity. This study investigates the parametric variation of droplet boiling regimes due to the experimental BCs (e.g surface roughness, ambient pressure) by performing separate effect tests employing high-speed visualization system. Differences in the impingement boiling characteristics of water droplets on solid (with surface roughness) and liquid metal (without surface roughness) in film boiling regime are investigated. A unique quantitative velocity data inside the droplet at several surface temperatures including (Leidenfrost temperatures) captured by Particle Tracking Velocimetry (PTV). This data is a unique component for the validation of CFD simulations which are performed to resolve the phenomena.

11:35AM H34.00006 Contribution of enhanced heat transfer in individual droplet impact cavities to overall heat transfer in spray cooling[1] , JOHN KUHLMAN, NICHOLAS HILLEN, West Virginia University, Mechanical & Aerospace Engineering Dept. — The thickness of the thin liquid film beneath the cavity formed by impact of an individual water droplet into a static liquid film over an unheated, horizontal surface (termed the sub-cavity liquid film thickness) was measured using a non-contacting optical thickness sensor, versus both time and radial distance away from the impact cavity centerline. Sub-cavity liquid film thickness data were obtained for ranges of Reynolds and Weber numbers expected for a commercial spray nozzle of interest, based on PDPA velocity and diameter data. These film thickness data were numerically integrated to determine the sub-cavity liquid volume time variation. The measured liquid film thickness decreases away from the cavity centerline over much of the cavity lifetime, for all test conditions. Computed sub-cavity volumes are typically between 50% to 100% of the droplet volume, and remain near this plateau value over much of the cavity lifetime. The measured sub-cavity liquid volume and cavity lifetime are used to estimate values for both the local cavity heat flux and the overall heat flux, averaged over the heater surface, that would be required to dry out the cavity prior to cavity fill in. These computed overall average heat flux values are compared with measured overall critical heat fluxes from the literature.

[1]Supported under NASA Cooperative Agreement NNX10AN0YA.

11:48AM H34.00007 Dropwise Condensation on a Radial Gradient Surface[1] , ASHLEY MACNER, SUSAN DANIEL, PAUL STEEN, Cornell University — In transient dropwise condensation from steam onto a cool surface, distributions of drops evolve by nucleation, growth, and coalescence. This study examines how surface functionalization affects drop growth and coalescence. Surfaces are treated by silanization to deliver either a spatially uniform contact-angle (hydrophilic, neutral, and hydrophobic) or a radial gradient of contact-angles. The time evolution of number-density and associated drop-size distributions are reported. For a typical condensation experiment on a uniform angle surface, the number-density curves show two regimes: an initial increase in number-density as a result of nucleation and a subsequent decrease in number-density as a result of larger scale coalescence events. Without a removal mechanism, the fractional coverage, regardless of treatment, approaches unity. For the same angle-surface, the associated drop-size distributions progress through four different shapes along the growth curve. In contrast, for a radial gradient surface where removal by sweeping occurs, the number-density increases and then levels off to a value close to the maximum number-density that is well below unity coverage and only two shapes of distributions are observed. Implications for heat transfer will be discussed.

[1]This work was supported by a NASA Office of the Chief Technologist's Space Technology Research Fellowship.

Monday, November 25, 2013 10:30AM - 12:40PM –

Session H35 Chaos, Fractals, and Dynamical Systems II: Analysis, Prediction, and Control 406 - George Haller, Eidgenossische Technische Hochschule Zurich

10:30AM H35.00001 Novel sampling strategies for dynamic mode decomposition[1] , JONATHAN H. TU, DIRK M. LUCHTENBURG, CLARENCE W. ROWLEY, Princeton University, STEVEN L. BRUNTON, J. NATHAN KUTZ, University of Washington — Originally introduced in the fluid mechanics community, dynamic mode decomposition (DMD) has emerged as a powerful tool for analyzing the dynamics of nonlinear systems. We present a theoretical framework that extends the understanding of DMD to nonsequential, potentially rank-deficient, time series and strengthens the connections between DMD and Koopman operator theory. This is in contrast to existing DMD theory, which deals primarily with sequential time series for which the snapshot (measurement) size is much larger than the number of snapshots (measurements). We demonstrate the benefits of applying DMD to nonsequential time series using two examples. First, we sample the simulated flow past a two-dimensional cylinder nonuniformly in time. The result is a more efficient DMD computation, with little effect on the accuracy of the dominant DMD modes and eigenvalues. Next, we combine particle image velocimetry data from multiple runs of a bluff-body wake experiment in a single DMD computation. This greatly mitigates the effects of noise and more clearly isolates the dominant modes.

[1]Funded by the AFOSR and the NSF

10:43AM H35.00002 Effects of small noise on the DMD/Koopman spectrum[1] , SHERVIN BAGHERI, Linne Flow Centre, KTH Mechanics, Stockholm — Koopman modes and Dynamic Mode Decomposition (DMD) have quickly become popular tools for extracting coherent structures associated with different frequencies from both (nonlinear) numerical and experimental flows. It is often expected (see Rowley et al, JFM, vol 641, 2009) that all the eigenvalues have zero growth rate (e.g. that they are located on the unit circle). However, in practice parabolic shapes and branches are observed in nearly all DMD spectra, and it is often the case that the tails of the parabolas are sensitive to the quality of the data set. In this talk, we provide a theoretical explanation for this parabolic form of the spectrum, and show that it arises due to the presence of noise. We show analytically that the presence of noise induces a damping on the eigenvalues, which increases quadratically with the frequency, and linearly with the non-normality of the linearized (Floquet) system. Thus the location of Koopman eigenvalues in the complex plane varies depending on the amount of noise in the environment, and one cannot expect any variant of the Dynamic Mode Decomposition algorithm to be fully robust to noise.

[1]Work supported by Swedish Research Council (VR-2010-3910)

10:56AM H35.00003 Attractor Identification from Empirical Data Using Diffusion-Mapped Delay Coordinates , ZRINKA GREGURIC FERENCEK, TYRUS BERRY, TIMOTHY SAUER, JOHN CRESSMAN, George Mason University — Nonlinear driven system can exhibit a diverse range of dynamics, from highly ordered to chaotic. These systems are ubiquitous, from atmospheric phenomena to brain function. In many cases, the governing equations for these systems are unknown. Here we present a dimensionality reduction algorithm based on diffusion-mapped delay coordinates that identifies the dimension and volume of the system's underlying attractor from empirical data. We generate data in the form of movies that are governed by the Rössler and Lorenz systems, as well as purely noisy and simple period dynamics. We show that this algorithm can be used to identify the dimensionality and volume of these attractors from empirical data. We then go on to apply this algorithm to a small electroconveting liquid crystals that supports multistable states that are characterized by patterns of creation, evolution, and annihilation of defects in the sample. We are able to identify the dimension and volume of their dynamics and use them to discriminate between these states.

11:09AM H35.00004 Nonlinear analysis of polymer electrolyte fuel cell dynamics with cathode two-phase flow , MICHAEL BURKHOLDER, SHAWN LITSTER, Department of Mechanical Engineering, Carnegie Mellon University — Water management in polymer electrolyte fuel cells (PEFCs) must be optimized to minimize parasitic costs. Removing water with excessive air flow rates at low-current, low-power conditions can be very parasitic, but these conditions can be unstable from the two-phase slug flow in the cathode air-delivery microchannels that occurs from the intrinsically low air and water flow rates. In this work, we use nonlinear analysis to understand the effect that varying currents and air flow rates have on PEFC dynamics. We estimate the dimension and entropy invariants indicative of dynamical complexity and stability from a reconstructed state space embedded using PEFC voltage data. We show that the estimated invariants can be correlated to the channel two-phase flow regime. We also investigate autocorrelation in the voltage signal by using diffusion analysis to estimate Hurst exponents. Lastly, we propose a reduced order map for application to real time PEFC water management.

11:22AM H35.00005 Is Chaotic Advection Inherent to Porous Media Flow? , DANIEL LESTER, GUY METCALFE, MIKE TREFRY, CSIRO — All porous media, including granular and packed media, fractured and open networks, are typified by the inherent topological complexity of the pore-space. This topological complexity admits a large number density of stagnation points under steady Stokes flow, which in turn generates a 3D fluid mechanical analouge of the Bakers map, termed the Baker's flow. We demonstrate that via this mechanism, chaotic advection at the pore-scale is inherent to almost all porous media under reasonable conditions, and such dynamics have significant implications for a range of fluid-borne processes including transport and mixing, chemical reactions and biological activity.

11:35AM H35.00006 Collaborative tracking and control in time-dependent stochastic dynamical systems[1] , ERIC FORGOSTON, Montclair State University, ANI HSIEH, Drexel University, IRA SCHWARTZ, U.S. Naval Research Laboratory, PHILIP YECKO, Montclair State University — 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. Lagrangian Coherent Structures (LCS) are found using collaborative tracking, and a control policy is formulated that utilizes knowledge of the LCS. The control strategy enables mobile sensors to autonomously maintain a desired distribution in the environment, and is evaluated with experimental data.

[1]Research supported by the Office of Naval Research.

11:48AM H35.00007 Nonlinear dynamic estimation with sparse sensors , STEVEN BRUNTON, University of Washington, JONATHAN TU, Princeton University, NATHAN KUTZ, University of Washington — We show that dimensionality reduction and compressive sensing strategies can be combined to estimate the state and/or parameters of a complex nonlinear system using only sparse measurements. L^2 based dimensionality reduction techniques, such as the proper orthogonal decomposition, are used to construct libraries spanning many dynamic phenomena, and sparse sensing is used to identify and reconstruct the dynamics from the library elements. This technique provides an objective and general framework for characterizing the underlying dynamics, stability, and bifurcations of the complex system. These methods are demonstrated on the complex Ginzburg-Landau equation using sparse, noisy measurements, as well as on the two-dimensional Navier-Stokes equation at low Reynolds number. Various spatiotemporal sampling strategies are investigated, with an emphasis on practical engineering considerations. We demonstrate that using a data-driven basis facilitates accurate nonlinear estimation from far fewer sensors than would typically be required of compressive sensing in a generic transform basis.

12:01PM H35.00008 Understanding the evolution of complex multiscale systems: Dynamic renormalization, non-equilibrium entropy and stochasticity , MARC PRADAS, MARKUS SCHMUCK, Department of Chemical Engineering, Imperial College London, UK, GRIGORIOS PAVLIOTIS, Department of Mathematics, Imperial Collge London, UK, SERAFIM KALLIADASIS, Department of Chemical Engineering, Imperial College London, UK — We present a novel methodology that enables the study the complex dynamics of dissipative systems. By means of a generic reduced equation which is also computationally efficient we tackle a fundamental problem in science: Many time-dependent problems are generally too complex to be fully resolved and hence some information needs to be neglected. A central question is then how can one systematically and reliably reduce the complexity of such high-dimensional systems without neglecting essential information. Popular examples of this are models for climate prediction, cell biology processes, or economics. We combine elements from nonlinear science, statistical physics, and information theory to develop a new stochastic strategy that rigorously shows how to replace the non-relevant degrees of freedom of an infinite-dimensional system by a finite random process statistically well defined [1]. A dynamic renormalization group approach reveals that the neglected information can be described in terms of an appropriately defined entropy for dissipative non-equilibrium processes which seems to have universal characteristics, thus providing a rational and systematic means for quantifying the evolution of dissipative systems.

[1] Schmuck, Pradas, Kalliadasis, Pavliotis. PRL 110, 244101(2013).

12:14PM H35.00009 Semi-automatic reduced order models from expert-defined transients[1] , ANDREAS CLASS, DENNIS PRILL, Areva Nuclear Professional School, Karlsruhe Institut of Technologie, Germany — Boiling water reactors (BWRs) not only show growing power oscillations at high-power low-flow conditions but also amplitude limited oscillations with temporal flow reversal. Methodologies, applicable in the non-linear regime, allow insight into the physical mechanisms behind BWR dynamics. The proposed methodology [1] exploits relevant simulation data computed by an expert choice of transient. Proper orthogonal modes are extracted and serve as Ansatz functions within a spectral approach, yielding a reduced order model (ROM). Required steps to achieve reliable and numerical stable ROMs are discussed, i.e. mean value handling, inner product choice, variational formulation of derivatives and boundary conditions.Two strongly non-linear systems are analyzed: The tubular reactor, including Arrhenius reaction and heat losses, yields sensitive response on transient boundary conditions. A simple natural convection loop is considered due to its dynamical similarities to BWRs. It exhibits bifurcations resulting in limit cycles. The presented POD-ROM methodology reproduces dynamics with a small number of spectral modes and reaches appreciable accuracy.

[1] Prill, D.& Class, A. Semi-automated POD-ROM non-linear analysis for future BWR stability analysis, Annals of Nuclear Energy, 20

[1]Funded by AREVA GmbH

12:27PM H35.00010 Cluster-based reduced-order modelling of a mixing layer[1] , EURIKA KAISER, BERND R. NOACK, LAURENT CORDIER, ANDREAS SPOHN, Institute PPRIME, France, MARC SEGOND, MARKUS ABEL, Ambrosys GmbH, Germany, GUILLAUME DAVILLER, CERFACS, France, ROBERT K. NIVEN, ADFA/UNSW, Australia — We propose a novel cluster-based reduced-order modelling (CROM) strategy of unsteady flows. CROM builds on the pioneering works of Gunzburger's group in cluster analysis (Burkardt et al. 2006) and Eckhardt's group in transition matrix models (Schneider et al. 2008) and constitutes a potential alternative to POD models. This strategy processes a time-resolving sequence of flow snapshots in three steps. First, the snapshot data is clustered into a small number of representative states in the phase space. The states are sorted by probability and transition considerations. Secondly, the transitions between the states are dynamically modelled via a Markov process. Finally, physical mechanisms are distilled by a refined analysis of the Markov process. The resulting CROM is applied to the Lorenz attractor as illustrating example and velocity fields of the spatially evolving incompressible mixing layer. For these examples, CROM is shown to distill non-trivial quasi-attractors and transitions processes. CROM has numerous potential applications for the systematic identification of physical mechanisms of complex dynamics, for comparison of flow evolution models, and for the identification of precursors for desirable and undesirable events.

[1]Partially funded by ANR Chair of Excellence TUCOROM, NSF PIRE grant OISE-0968313, and EC's Marie-Curie ITN program.

Monday, November 25, 2013 10:30AM - 12:01PM –

Session H36 Geophysical: Oceanographic V 407 - Roberto Verzicco, University of Rome, Tor Vergata

10:30AM H36.00001 Relevancy of the buoyancy Reynolds number in stably stratified turbulence[1] , BENJAMIN MATER, SUBHAS KARAN VENAYAGAMOORTHY, Colorado State University — The buoyancy Reynolds number, $Re_b = \epsilon/(\nu N^2)$, has become a widely popular parameter with which to describe turbulent mixing in the stratified environment of the open ocean. This popularity has arisen largely on the practical grounds that the constituent quantities are available through common measurement techniques: estimates of turbulent kinetic energy dissipation (ϵ) are available from observations of fine-scale shear, and the buoyancy frequency (N) can be determined from profiles of density. Despite practical appeal, however, Re_b is ambiguous in that it fails to distinguish between regimes of weak stratification and strong turbulence. This becomes obvious in the formulation $Re_b = Re_L(Fr_k)^2$, where $Re_L = k^2/(\epsilon\nu)$ is a turbulent Reynolds number, $Fr_k = \epsilon/(Nk)$ is a turbulent Froude number, and k is the turbulent kinetic energy. In considering both Re_L and Fr_k independently, the time scale of the turbulence, $T_L = k/\epsilon$, is made explicit. We explore the duality of Re_b in describing mixing efficiency using a $Re_L - Fr_k$ parameter space and argue the importance of T_L in parameterization of flow behavior. Data from direct numerical simulations, laboratory experiments, and field observations are considered.

[1]This work funded by the Office of Naval Research

10:43AM H36.00002 Stability of Baroclinic Vortices in Rotating Stratified Flows , MANI MAHDINIA, UC Berkeley, PEDRAM HASSANZADEH, Harvard University, PHILIP MARCUS, UC Berkeley — The stability of axisymmetric shielded vortices in rotating linearly stratified flows is examined using three-dimensional numerical simulations of the Boussinesq equations. Vortices are initially in cyclo-geostrophic/hydrostatic dissipationless equilibrium in accord with the constraint presented in Hassanzadeh et al. (2012 JFM) and Aubert et al. (2012 JFM). For a range of Rossby numbers, vortex aspect ratios, and Burger numbers relevant to the oceanic and atmospheric flows, both cyclones and anticyclones are studied, and the type and growth rate of instabilities are reported for the unstable cases. Significance of these results for the cyclone-anticyclone asymmetry observed in rotating stratified flows is discussed.

10:56AM H36.00003 From Balanced Barotropic and Baroclinic Shear to Turbulence in Rotating and Stratified Flow , ERIC AROBONE, SUTANU SARKAR, University of California, San Diego — In the oceanic submesoscale regime, rotation is important but does not control dynamics. Instabilities and nonlinear cascades are possible even for stably stratified flows. Previous work by the authors explored the rotating stratified barotropic shear layer. Here, the vertical wavenumber band associated with linear barotropic instability greatly increased when centerline absolute vorticity was approximately zero. Correspondingly, nonlinear simulations showed a marked transition event during the changing in sign of centerline absolute vorticity. Our study will include direct numerical simulations exploring the effect of a weak isolated front in combination with the previously explored barotropic shear layer. The primary aim of this study is to explore how the instabilities of the barotropic simulations are modified by weak baroclinicity. We will explore coherent structure evolution to qualitatively assess the importance of the numerous possible instability mechanisms, especially the zero absolute vorticity mechanism. Additionally, energy and enstrophy budgets will be analyzed comparing the various pathways from large-scale kinetic and potential energies to turbulence.

11:09AM H36.00004 Efficiency of mixing by heating or cooling in thermally stratified nonlinear spin-up[1] , MELINE BAGHDASARIAN, ARTURO PACHECO-VEGA, California State University, Los Angeles, ROBERTO VERZICCO, Universita di Roma "Tor Vergata", J. RAFAEL PACHECO, SAP Americas — Spin-up (the transient flow of a fluid, either at rest or in solid body rotation, due to an increase in rotation rate), is particularly relevant to large-scale geophysical flows. Here we present numerical experiments of spin-up in a cylindrical container with shear-free upper boundary for four different thermal boundary conditions on the horizontal walls: (1) prescribed temperatures, (2) adiabatic conditions, (3) adiabatic on top and prescribed temperature on the bottom, and (4) prescribed temperature on top and adiabatic on the bottom. Studies on spin-up subjected to different boundary conditions and stratifications matter, as they may be helpful to understand the spin of water masses in basins for different physical scenarios. Most of the time, new water masses are formed at the surface by cooling, and their spin-up is clearly of utility in determining ensuing flow patterns. The focus here is on the efficiency of mixing due to spin-up when the horizontal boundaries are subjected to different thermal conditions.

[1]This work has been supported by NSF HRD-0932421.

11:22AM H36.00005 Turbulence Statistics in the Inner Part of the Coastal Ocean Bottom Boundary Layer[1] , ADITYA NAYAK, CHENG LI, BOBAK KIANI, JOSEPH KATZ, Johns Hopkins University — PIV measurements were performed in the inner part of the coastal bottom boundary layer under varying bottom roughness conditions, relative wave current orientation and ratio of mean current to amplitude of wave induced motion (WCR). Velocity distributions with resolution of 4.5 mm were obtained in two 28 x 28 cm^2 planes down to 5 mm off the seabed. Co-located ADV measurements were used to calculate Reynolds stresses by filtering out wave-induced motions from PIV data, and high-resolution sonar was used to map the bottom roughness. Mean velocity and Reynolds stress profiles varied with WCR and their relative alignment. An inflection in mean velocity profile developed below the log layer for WCR$\sim$1, but not for higher ratios. Reynolds stresses peaked in the lower portion of the log layer, decreasing with elevation above. The peak location, and the stress scaling trends depended on WCR. A second stress peak appeared just above the ripple crest. Wave-induced wall-normal momentum transport ("stress") became substantial within the roughness sublayer, where the relative phase between streamwise and vertical velocity components were altered. The dissipation rate profiles showed a rapid increase with decreasing elevation, but the shear production varied with WCR and roughness orientation.

[1]Sponsored by NSF - OCE.

11:35AM H36.00006 Numerical Simulation of Bottom Boundary Layer Turbulence under an Internal Solitary Wave of Depression[1] , TAKAHIRO SAKAI, University of Southern California, PETER DIAMESSIS, Cornell University, GUSTAAF JACOBS, San Diego State U. — The turbulent bottom boundary layer (BBL) under a mode-1 internal solitary wave (ISW) of depression is examined using spectral multidomain-based implicit large eddy simulations. The ISW propagates in a two-layer stratification in either quiescent waters or against an idealized barotropic current augmented with an idealized laminar Blasius boundary layer. Various non-trivial aspects of this highly expensive numerical process study are examined with the focus on the effectiveness of various strategies aimed towards establishing a self-sustained near-bed turbulent wake. Such a near-bed wake is considered to be the 3-D extension of the corresponding 2-D global instability observed in the ISW footprint by previous investigations. To this end, in analogy with recent aerodynamics studies, the incorporation of localized volumetric forcing aft the separated BBL under the ISW appears to be the optimal choice. Following a characterization of the structure and dynamics of the ISW-induced BBL, we discuss key differences with related laboratory experiments, assess the relevance of volumetric forcing to both the laboratory and field and speculate on whether numerically simulated self-sustained near-bed turbulence is indeed possible for a pure no-slip bottom.

[1]Support by grants ONR N00014-11-1-0511 and NSF OCE-0845558

11:48AM H36.00007 Two-dimensional irrotational nonlinear flow over arbitrary bottom topography in a Channel , SRIKUMAR PANDA, SUBASH CHANDRA MARTHA, Indian Institute of Technology Ropar, India, ALOKNATH CHAKRABARTI, Indian Institute of Science, Bangalore, India — The problem involving two-dimensional irrotational flow over arbitrary bottom topography in an infinite channel is investigated within the framework of fully nonlinear theory. This two-dimensional problem can be cast into a mixed boundary value problem. Using certain transformations, the mixed boundary value problem is formulated as a Dirichlet problem. The Dirichlet problem is solved by the aid of integral equation method, and the solution of the Dirichlet problem involves an unknown potential function that is to be determined. This unknown function can be determined completely, once a pair of singular integral equations appearing here are solved completely. By the help of Newton's method, the free surface profile is determined and shown graphically.

Monday, November 25, 2013 2:00PM - 2:35PM –

Session J27 Invited Session: Do Swimming Animals Mix the Ocean? Spirit of Pittsburgh Ballroom A - Morteza Gharib, California Institute of Technology

2:00PM J27.00001 Do swimming animals mix the ocean?[1] , JOHN DABIRI, California Institute of Technology — Perhaps. The oceans are teeming with billions of swimming organisms, from bacteria to blue whales. Current research efforts in biological oceanography typically focus on the impact of the marine environment on the organisms within. We ask the opposite question: can organisms in the ocean, especially those that migrate vertically every day and regionally every year, change the physical structure of the water column? The answer has potentially important implications for ecological models at local scale and climate modeling at global scales. This talk will introduce the still-controversial prospect of biogenic ocean mixing, beginning with evidence from measurements in the field. More recent laboratory-scale experiments, in which we create controlled vertical migrations of plankton aggregations using laser signaling, provide initial clues toward a mechanism to achieve efficient mixing at scales larger than the individual organisms. These results are compared and contrasted with theoretical models, and they highlight promising avenues for future research in this area.

[1]Funding from the Office of Naval Research and the National Science Foundation is gratefully acknowledged.

Monday, November 25, 2013 2:00PM - 2:35PM –

Session J28 Invited Session: Dynamics of Transient Liquid Injection Spirit of Pittsburgh Ballroom B/C - Forman Williams, University of California, San Diego

2:00PM J28.00001 Dynamics of Transient Liquid Injection[1] , WILLIAM SIRIGNANO, University of California, Irvine — Start-up transients and steady injection of liquid through round and annular orifices into ambient gas are examined. Various relevant hydrodynamic instabilities are examined together with their synergisms; capillary instability; vortex-ring (Widnall) instability; Kelvin-Helmholtz (i.e., shear-driven) instability; and Rayleigh-Taylor instability. Studies of both full jets and liquid segments of the jet are considered. Different instabilities and synergisms of instabilities produce different wavelengths on the jet interface. Surface waves, cone and sheet formations, and liquid-sheet tearing are examined. The breakdown of axisymmetry is related to the various instabilities. Accelerations of both the exit jet velocity and the liquid in the cones and ligaments are examined for Rayleigh-Taylor instabilities. Identification is sought of the length scales related to eventual breakup and droplet or ligament formation. Cavitation at high pressures is explained. Bubble growth and collapse in the internal orifice flow are discussed. Numerical problems with prediction of liquid stream breaking are discussed. Analysis and computations are emphasized but some experiments are discussed.

[1]Support from Army Research Office is acknowledged.

Monday, November 25, 2013 2:40PM - 3:15PM –

Session K27 Invited Session: Porous-medium Convection: New Problems from CO2 Sequestration Spirit of Pittsburgh Ballroom A - George Homsy, University of California, Santa Barbara

2:40PM K27.00001 Porous-medium convection: new problems from CO2 sequestration , JOHN LISTER, University of Cambridge — Large scale injection and storage of supercritical carbon dioxide (CO2) into deep saline aquifers is proposed to offset anthropogenic emissions and mitigate climate change. Many aspects of the resultant porous flows provoke fundamental fluid-mechanical problems. The rise and spread of the buoyant CO2 plume beneath an overlying impermeable stratum is a classic gravity current, but with the undesirable extra possibility of upward leakage through fractures. Fortunately, long-term trapping mechanisms exist. One such, dissolution of CO2 into the underlying brine, produces a denser solution which thus convects reassuringly downwards. Consideration of the convective flux prompts re-examination of high-Ra convection in a porous medium, which is found to have a strikingly different asymptotic form from that in a pure fluid. The high-Ra regime of Rayleigh-Darcy convection has an ordered interior with a linear mean temperature gradient and a superposed vertical columnar heat-exchanger flow whose wavelength is consistent with the $Ra^{-5/14}$ scaling predicted by an asymptotic stability analysis. Quantification of the convective dissolution flux allows evolution towards saturation in confined aquifers, or the erosion of a gravity current in open aquifers, to be calculated.

Monday, November 25, 2013 2:40PM - 3:15PM –

Session K28 Invited Session: Drag Reduction and the Dynamics of Turbulence in Simple and Complex Fluids Spirit of Pittsburgh Ballroom B/C - Gareth H. McKinley, Massachusetts Institute of Technology

2:40PM K28.00001 Drag reduction and the dynamics of turbulence in simple and complex fluids[1] , MICHAEL GRAHAM, University of Wisconsin-Madison — Addition of a small amount of very large polymer molecules or micelle-forming surfactants to a liquid can dramatically reduce the energy dissipation it exhibits in the turbulent flow regime. This rheological drag reduction phenomenon is widely used, for example in the Alaska pipeline, but it is not well-understood, and no comparable technology exists to reduce turbulent energy consumption in flows of gases, in which polymers or surfactants cannot be dissolved. The most striking feature of this phenomenon is the existence of a so-called maximum drag reduction (MDR) asymptote: for a given geometry and driving force, there is a maximum level of drag reduction that can be achieved through addition of polymers. Changing the concentration, molecular weight or even the chemical structure of the additives has no effect on this asymptotic value. This universality is the major puzzle of drag reduction. We describe direct numerical simulations of turbulent channel flow of Newtonian fluids and viscoelastic polymer solutions. Even in the absence of polymers, we show that there are intervals of "hibernating" turbulence that display very low drag as well as many other features of the MDR asymptote observed in polymer solutions. As viscoelasticity increases, the frequency of these intervals also increases, leading to flows that increasingly resemble MDR. A simple theory captures key features of the intermittent dynamics observed in the simulations. Additionally, simulations of "edge states," dynamical trajectories that lie on the boundary between turbulent and laminar flow, display characteristics that are similar to those of hibernating turbulence and thus to the MDR asymptote, again even in the absence of polymer additives. Based on these observations, we propose a tentative unified description of rheological drag reduction. The existence of "MDR-like" intervals even in the absence of additives sheds light on the observed universality of MDR and may ultimately lead to new flow control approaches for improving energy efficiency in a wide range of processes.

[1]This work was supported by the National Science Foundation through grants CBET-0730006 and CBET-1066223 as well as the Air Force Office of Scientific Research through grant FA9550-11-1-0094 (Flow Interactions and Control Program).

Monday, November 25, 2013 3:35PM - 5:32PM –

Session L1 Geophysical: Oceanographic VI 323 - Patrice Le Gal, Aix Marseille University

3:35PM L1.00001 Breaking of the internal tide , KARL HELFRICH, Woods Hole Oceanographic Institution, ROGER GRIMSHAW, Loughborough University, EDWARD JOHNSON, University College London — Nonlinear steepening of low-mode internal tides and the subsequent arrest of steepening by non-hydrostatic dispersion is a common mechanism for the generation of internal solitary waves in the ocean. However, it is known that the earth's rotation may interfere and prevent the emergence of the solitary waves. The Ostrovsky equation, the Korteweg-de Vries equation with a nonlocal integral term representing the effects of rotation, is introduced as model for these processes. Recent work on a breaking criteria for the reduced Ostrovsky equation (in which the linear non-hydrostatic dispersive term with a third-order derivative is eliminated) is discussed. This equation is integrable provided a certain curvature constraint is satisfied. It is demonstrated, through theoretical analysis and numerical simulations, that when this curvature constraint is not satisfied at the initial time, then wave breaking inevitably occurs. The breaking criteria is applied to several oceanic examples including internal tides in the South China Sea and radiation of the internal tide from the Hawaiian Island chain.

3:48PM L1.00002 From weak to strong turbulence: a traveling wave tour , FRANCESCO FEDELE, Georgia Institute of Technology — The weak wave turbulence of Zakharov unveiled the dynamics of ocean waves as that of a sea of nonlinearly interacting dispersive elementary waves. Their dispersive properties and energy cascade can be observed and measured in the ocean. In this regard, I will discuss recent experiments off the Venice coast that exploit a Variational Wave Acquisition Stereo System (VWASS) to study the space-time dynamics of sea waves [Fedele et al. 2013, Ocean Modeling]. The delicate balance of dispersion and nonlinearities may yield the formation of solitons or traveling waves [Fedele & Dutykh 2012, JFM 712:646]. These are introduced in the context of the Euler equations and the associated third order compact Zakharov equation. Traveling waves exist also in the strong turbulence of the Navier-Stokes (NS) equations. Indeed, for bounded geometries I will show that the NS equations can be reduced to generalized Camassa-Holm equations [Fedele 2012, Fluid Dyn. Res. 44:045509; Fedele & Dutykh 2013, EPL 101:34003]. From a dynamical system perspective, in phase space the associated vector field supports an invariant group orbit manifold, which corresponds in physical space to smooth and singular axisymmetric vortexons [5].

4:01PM L1.00003 Large overturns at a model Luzon Strait topography: an application of the Immersed Boundary Method , NARSIMHA RAPAKA, SUTANU SARKAR, University of California San Diego — An Immersed Boundary Method (IBM) is used to study the internal wave field in a model of the Luzon Strait topography, a double ridge system with different heights and local roughness. Scale compression is employed, leading to horizontal scales of O (100 m) instead of km. Strong internal tide beams are generated on both the east and the west ridges near the critical regions where the internal wave characteristic slope matches with that of the topographic slope. A large fraction of the radiated tidal energy is confined near the top surface owing to ducting by the pycnocline. Isopycnal displacement is particularly large (around 200 m after scaling up) at the eastern flank of the west ridge, which corresponds to the station N2 of Alford et al., 2011 (JPO). The wave displacements at the east ridge is influenced by the proximity of the pycnocline. The spatial distribution of baroclinic energy conversion and internal wave flux is assessed.

4:14PM L1.00004 Response of Ocean Circulation to Different Wind Forcing in Puerto Rico and US Virgin Islands[1] , MIGUEL SOLANO, EDGARDO GARCIA, Universidad de Puerto Rico, STAFANO LEONARDI, University of Texas at Dallas, MIGUEL CANALS, JORGE CAPELLA, Universidad de Puerto Rico — The response of the ocean circulation to various wind forcing products has been studied using the Regional Ocean Modeling System. The computational domain includes the main islands of Puerto Rico, Saint John and Saint Thomas, located on the continental shelf dividing the Caribbean Sea and the Atlantic Ocean. Data for wind forcing is provided by an anemometer located in a moored buoy, the Coupled Ocean-Atmosphere Mesoscale Prediction System (COAMPS) model and the National Digital Forecast Database (NDFD). Hindcast simulations have been validated using hydrographic data at different locations in the area of study. Three cases are compared to quantify the impact of high resolution wind forcing on the ocean circulation and the vertical structure of salinity, temperature and velocity. In the first case a constant wind velocity field is used to force the model as measured by an anemometer on top of a buoy. In the second case, a forcing field provided by the Navy's COAMPS model is used and in the third case, winds are taken from NDFD in collaboration with the National Centers for Environmental Prediction. Validated results of ocean currents against data from Acoustic Doppler Current Profilers at different locations show better agreement using high resolution wind data as expected.

[1]Thanks to CariCOOS and NOAA

4:27PM L1.00005 Relation of Lagragian structures and drifter dynamics in the Gulf of Mexico[1]

, CAROLINA MENDOZA, Universidad Politecnica de Madrid, ANA MARIA MANCHO, ICMAT, CSIC, STEPHEN WIGGINS, School of Mathematicss, University of Bristol — We use a Lagrangian descriptor (the so called function M) which measures the length of particle trajectories on the ocean surface over a given interval of time [1,2,3]. With this tool we identify the Lagrangian skeleton of the flow and compare it on three datasets over the Gulf of Mexico during the year 2010. The satellite altimetry data used come from AVISO and simulations from HYCOM GOMl0.04 experiments 30.1 and 31.0. We contrast the Lagrangian structure and transport using the evolution of several surface drifters. We show that the agreement in relevant cases between Lagrangian structures and dynamics of drifters depends on the quality of the data on the studied area.

[1] C. Mendoza, A.M. Mancho. Phys. Rev. Lett. 105 (2010), 3, 038501.
[2] C. Mendoza, A.M. Mancho. Nonlin. Proc. Geophys. 19 (4) (2012) 449-472.
[3] A.M. Mancho, S. Wiggins, J. Curbelo, C. Mendoza. Commun. Nonlinear. Sci. Numer. Simul. 18 (2013) 3530-3557.

[1]We acknowledge to the grants: Becas de Movilidad de Caja Madrid 2011, MTM2011-26696, ILINK-0145, ONR Grant Number N00014-01-1-0769 and ICMAT Severo Ochoa SEV-2011-0087

4:40PM L1.00006 ABSTRACT WITHDRAWN –

4:53PM L1.00007 Laboratory experiments investigating the influence of subglacial discharge on submarine melting of Greenland's Glaciers[1]

, CLAUDIA CENEDESE, Woods Hole Oceanographic Institution — A set of idealized laboratory experiments investigates the ice-ocean boundary dynamics near a vertical "glacier" (i.e. no floating ice tongue) in a two-layer stratified fluid, similar to Sermilik Fjord where Helheim Glacier terminates. In summer, the discharge of surface runoff at the base of the glacier (subglacial discharge) causes the circulation near the glacier to be much more vigorous and is associated with a larger melt rate than in winter. In the laboratory the effect of a subglacial discharge is simulated by introducing fresh water at melting temperatures from a source at the base of the ice block representing the glacier. The influence of both a line and a point source of subglacial discharge on submarine melting are investigated. A buoyant plume of cold melt water and subglacial discharge water entrains ambient waters and rises vertically until it finds either the interface between the two layers or the free surface. The results suggest that the melt water deposits within the interior of the water column and not entirely at the free surface, as confirmed by field observations and numerical experiments. Furthermore, the submarine melting increases with subglacial discharge. Finally, a non-monotonic dependence of the submarine melting on the distance between two point sources of subglacial discharge suggests that the distribution and number of sources of subglacial discharge may play an important role in glacial melt rates.

[1]Support was given by the National Science Foundation project OCE-113008 and the WHOI-Arctic Research Initiative.

5:06PM L1.00008 Upstream versus downstream control of meltwater plumes under ice shelves

, ANDREW WELLS, University of Oxford — In many locations the Greenland and Antarctic ice sheets discharge into the ocean through ice shelves floating on top of a warm salty ocean. The turbulent buoyancy-driven flow of meltwater beneath the sloping ice-shelf base enhances heat transfer and provides a feedback on ice melting rates, with consequences for ice sheet dynamics and predictions of sea-level rise. Previous steady-state models of meltwater plumes under ice shelves have solved for the development of flow along the slope from an initial source, corresponding to solely upstream control of the plume dynamics. I re-interpret the plume dynamics embedded within the framework of a time-dependent model, and show that the flow exhibits distinct regimes depending on the source conditions. Solutions with upstream control are physically consistent for certain source conditions, but the plume is influenced by a combination of upstream and downstream conditions in other regions of parameter space. The dynamics are illustrated for flow underneath a two-dimensional ice shelf of initially constant basal slope, and stable attracting states are determined. The implications for modelling meltwater flow under ice shelves are discussed.

5:19PM L1.00009 Turbulent mixing in a barrier layer

, HIEU PHAM, SUTANU SARKAR, University of California, San Diego — Large-eddy simulation (LES) is used to investigate the erosion of a barrier layer in the upper ocean by wind-driven turbulence. The initial vertical density profile consists of three regions: a buoyantly neutral surface layer, an isothermal salt-stratified region (the so-called barrier layer) and a thermally stratified deep region. A constant wind-stress and a diurnal heat flux are applied at the surface to drive the turbulence. During the night, the wind stress generates shear in the mixed layer. The shear deepens and causes Holmboe shear instability at the interface between the mixed layer and the barrier layer where the gradient Richardson number falls below 0.25. In time, the barrier layer becomes thinner, and the mixed layer thickens with increasing surface salinity. In the morning, as the surface heat flux warms the mixed layer, a temperature inversion is formed on top of the barrier layer. The surface heating suppresses the turbulence in the surface layer; however, the mean shear continues to increase and causes occasional bursts of shear instability. The instability mixes up the temperature inversion by the afternoon.

Monday, November 25, 2013 3:35PM - 5:58PM –

Session L2 Convection and Buoyancy-Driven Flows V: Binary Systems and Stratified Flows 324

- Edgar Knobloch, University of California, Berkeley

3:35PM L2.00001 Thermal Stratification by Steam Condensation of RCIC in Suppression Pool

, KOJI OKAMOTO, DAEHUN SONG, NEJDET ERKAN, The University of Tokyo — In Fukushima-Daiichi NPP accident, the RCIC operated more than a couple of days. The steam condensation at the supression pool may cause the thermal-stratification. The phenomena may affects on the capacity of RCIC and also the progression of the event in case of Severe Accident. To investigate the mechanism of formation of thermal stratification and the effects in suppression pool, down-sized SP model was designed and time resolved temperature and pressure data were acquired. During the experiments well-established stationary thermal stratification was detected since the start of steam injection. PIV was carried out to figure out the natural circulation due to the buoyancy and the mixing interface, which decides the pressure suppression capacity of suppression pool (SP). Numerical simulation was carried out with ANSYS CFX 14.0 in single phase and validated with experimental data.

3:48PM L2.00002 Chemical control of hydrodynamic instabilities in partially miscible two-layer systems

, A. DE WIT, L.A. RIOLFO, L. LEMAIGRE, Université Libre de Bruxelles, F. ROSSI, University of Salerno, M. RUSTICI, M.A. BUDRONI, Universita di Sassari — Hydrodynamic instabilities at the interface between two partially miscible liquids impact numerous applications including sequestration of supercritical liquid CO_2 in old petroleum reservoirs or saline aquifers. As an alternative to difficult *in situ* studies of the related mixing dynamics, we introduce a new laboratory-scale model system on which buoyancy- and Marangoni-driven convective instabilities of partially miscible two-layer systems can easily be studied and controlled in presence or not of chemical reactions. This system consists in the stratification of a pure ester on top of a denser partially miscible aqueous solution in the gravitational field. The rich convective dynamics observed upon partial dissolution of the ester in the water followed by its hydrolysis highlight the specificity of partially miscible systems as compared to fully miscible or immiscible ones, i.e. the possibility to control the convective pattern and the mixing properties by tuning (i) the intrinsic miscibility of the ester in water, (ii) the feedback of the dissolved species on its own miscibility as well as (iii) the composition and reactivity of the aqueous solution with the ester phase.

4:01PM L2.00003 Convective dissolution of carbon dioxide in salted water: linear stability analysis and effect of control parameters , VANESSA LOODTS, LAURENCE RONGY, ANNE DE WIT, Non Linear Physical Chemistry Unit, Université Libre de Bruxelles (ULB) — We study the convective dissolution of carbon dioxide (CO_2) in salted water theoretically. We perform a linear stability analysis with regard to buoyancy-driven convection of the time-dependent concentration profiles of CO_2 diffusing into the aqueous solution. On the basis of a parameter-free dimensionless model, we predict the time of onset and wavenumber of the instability when the system becomes unstable, i.e. when the growth rate of the instability equals zero. We also define a characteristic growth rate σ^* quantifying the growth of the perturbations in the unstable regime. We find good agreement of σ^* with growth rates of buoyancy-driven fingering available in the literature. We moreover explicit the procedure to compare the dimensionless theoretical prediction with dimensional experimental and numerical data and analyze the influence of parameters controlling implicitly the characteristic length and time scales of the problem. We find that increasing the partial pressure of CO_2, or decreasing the aqueous salt concentration or the temperature destabilize the system, leading to faster growing buoyancy-driven fingers.

4:14PM L2.00004 Convection in binary fluids with phase change: solutocapillarity, thermocapillarity and buoyancy[1] , YAOFA LI, MINAMI YODA, Georgia Institute of Technology — Evaporative cooling is of interest in thermal management applications. In most cases, thermocapillary stresses drive liquid coolant away from hot regions, adversely affecting performance. Volatile binary fluids can, however, be tailored with *solutocapillary* stresses that drive liquid instead towards hot regions. Although such binary-fluid coolants could improve the cooling performance of devices such as heat pipes, convection in a binary fluid subject to phase change, especially in a confined geometry in the (near-)absence of non-condensables as is the case in heat pipes, is poorly understood. Capillary-buoyancy convection in liquid layers (with depths of a few mm) driven by temperature differences as great as 10 °C over a horizontal distance of 4.9 cm was therefore studied with particle-image velocimetry (PIV). The flow of water-methanol mixtures (with methanol fractions as great as 60%) was studied under conditions where the vapor space was filled with ambient air, and a mixture of water and methanol vapor with a small amount of air. The results show that varying the amount of air in the vapor space has a marked effect on the flow in the liquid layer.

[1]Supported by ONR

4:27PM L2.00005 Compositional transport in solidifying aqueous binary solution , JIN-QIANG ZHONG, ZUO-CHAO YIN, Tongji University, Shanghai, China, QIWEI XUE, Yale University, New Haven, CT, USA, JOHN WETTLAUFER, Yale University, New Haven, CT, USA and Oxford University, Oxford, UK — We observe the formation of double-diffusive layers adjacent to mushy layers that form during the directional solidification of aqueous ammonium chloride. The plumes emerging from chimney's in the mushy layers continuously supply a buoyancy flux in the (finite) liquid region above, driving downward motion of double-diffusive layers. The downward velocity of the layers is found to be in good agreement with a filling box model that captures the crucial hydrodynamics of the entraining buoyant plumes and compositional transport. We demonstrate that the buoyancy flux through the system decays according to a similarity solution. We note that the experimental findings provide some insight into the brine transport in growing sea ice.

4:40PM L2.00006 Colliding Convectons , EDGAR KNOBLOCH, University of California, Berkeley, ISABEL MERCADER, ORIOL BATISTE, ARANTXA ALONSO, UPC, Barcelona, Spain — Convectons are strongly nonlinear spatially localized states found in thermally driven fluid flows. In systems with midplane reflection symmetry stationary convectons of odd and even parity lie on a pair of intertwined branches that form the backbone of the snakes-and-ladders structure of a "pinning" region in parameter space (Mercader et al., J. Fluid Mech. 667 (2011) 586). When the midplane reflection symmetry is broken, the odd parity convectons start to drift with a speed that depends on the magnitude of the symmetry-breaking and the convecton length. Direct numerical simulations are used to study head-on and follow-on collisions between such drifting convectons in binary fluid convection, and the results compared and contrasted with corresponding dynamics in a Swift-Hohenberg model studied by Houghton and Knobloch (PRE 84 (2011) 016204). In contrast to completely integrable systems the collisions are strongly inelastic (Mercader et al., J. Fluid Mech. 722 (2013) 240).

4:53PM L2.00007 Accuracy of the 2D+t Approximation for Turbulent Wakes in Stratified Flows , LAURA PAULEY, Pennsylvania State University — Wakes in the ocean can be produced by a stationary object in a current or by a moving object in stationary water. When viewed in a reference frame moving with the object, the wake can persist thousands of object diameters downstream. Due to the extensive domain, an unsteady two-dimensional (2D+t) computation is often used to sweep downstream through the wake development. The 2D+t computation approximates the development of the wake at a fixed location as an object moves past but applies cyclical boundary conditions in the streamwise direction. A Parabolized Navier-Stokes (PNS) method has the same numerical efficiency as the 2D+t method but includes additional streamwise gradient terms found in the three-dimensional governing equations. The present paper investigates the accuracy of the 2D+t approximation for stratified turbulent wake flows for a range of Froude numbers and Reynolds numbers. The 2D+t results are compared with results from 3D Navier-Stokes computations and results from PNS computations to identify criteria at which the 2D+t method will yield accurate results.

5:06PM L2.00008 Large eddy simulation of buoyancy induced asymmetry in horizontal jets , NIRANJAN GHAISAS, STEVEN FRANKEL, School of Mechanical Engineering, Purdue University — Horizontal injection of a heavier fluid into a lighter ambient leads to a horizontal buoyant jet. This configuration is marked by the simultaneous presence of stable stratification above the jet centerline, and unstable stratification below it. This leads to unequal rates of turbulent mixing and an asymmetric development of the jet above and below the centerline. This asymmetry between the stably stratified and unstably stratified regions in the horizontal jet is investigated using large eddy simulations in this study. Parameters such as radial half-widths and measures of anisotropy are investigated. Differences in the structures occurring in stable and unstable stratifications are pointed out. Finally, a dynamic mode decomposition analysis is performed with the stably stratified and unstably stratified regions considered together, as well as individually. It is seen that the unstably stratified region is more energetic, and prone to instabilities, as compared to the stably stratified region.

5:19PM L2.00009 A periodic mixing mechanism in stratified turbulent Taylor-Couette flow , ROSALIND OGLETHORPE, C.P. CAULFIELD, BP Institute & DAMTP, University of Cambridge, ANDREW W. WOODS, BP Institute, University of Cambridge — We present results from a series of laboratory experiments to study the mixing mechanism in two-layer, stratified turbulent Taylor-Couette flow. We focus on the case of strong stratification, where the density difference $\Delta\rho$ is sufficiently high that the vertical buoyancy flux across the interface is constant (as found by Woods *et al.* (2010)JFM**663** and Oglethorpe *et al.* (2013)JFM**721**). We vary the radius, R_1, surface roughness and rotation rate, Ω, of the inner cylinder, relative to the stationary outer cylinder, of radius R_2. The measurements of the density field near the interface, using both conductivity probe data and visualization techniques, show a periodic signal which is associated with the mixing. We find that the period of the signal is given by $T \propto (2\pi/\Omega)(R_2/R_1)$. We also find that the mean angular momentum in the bulk of the flow is constant, and depends on the surface roughness of the inner cylinder. We use these results to present an interpretation of the mixing mechanism related to the periodic signal.

5:32PM L2.00010 Flows and Stratification of an Enclosure Containing Both Localised and Vertically Distributed Sources of Buoyancy[1] , JAMIE PARTRIDGE, PAUL LINDEN, University of Cambridge — We examine the flows and stratification established in a naturally ventilated enclosure containing both a localised and vertically distributed source of buoyancy. The enclosure is ventilated through upper and lower openings which connect the space to an external ambient. Small scale laboratory experiments were carried out with water as the working medium and buoyancy being driven directly by temperature differences. A point source plume gave localised heating while the distributed source was driven by a controllable heater mat located in the side wall of the enclosure. The transient temperatures, as well as steady state temperature profiles, were recorded and are reported here. The temperature profiles inside the enclosure were found to be dependent on the effective opening area A^*, a combination of the upper and lower openings, and the ratio of buoyancy fluxes from the distributed and localised source $\Psi = \frac{B_w}{B_p}$.

[1]Industrial CASE award with ARUP

5:45PM L2.00011 The Radial Spreading of Intrusions Originating from a Plume in Stratified Fluid , TAMAR RICHARDS, University of Alberta, QUENTIN AUBOURG[1], University of Grenoble, BRUCE SUTHERLAND, University of Alberta — Supervolcanoes send a plume of hot particle-laden air into the stratosphere where it eventually falls back upon itself as a fountain and then spreads laterally at its neutral buoyancy level. In order to gain insight into the initial spreading of such intrusions, we have performed laboratory experiments of fresh water injected downward through a turbulent plume nozzle into a uniformly stratified fluid. Though neglecting the influence of particles and anelastic effects, the experiment provides insight into the dynamics of radially spreading intrusions in the buoyancy-inertia regime within the Boussinesq approximation. Our theoretical and experimental results extend the prediction of Bloomfield and Kerr (JFM 1998,2000) to predict the spreading height as a function of the source momentum, buoyancy and ambient stratification for buoyancy- as well as momentum-driven sources. We find the radius of the front increases as a power law with approximately $3/4$ exponent, different from self-similarity theory, which predicts a $2/3$ exponent. Nonetheless, the intrusion structure adopts a self-similar shape with scaled height as a function of scaled radius having an approximate $1/2$ power law from nose to tail.

[1]This research was performed while on an internship at the University of Alberta

Monday, November 25, 2013 3:35PM - 5:58PM –

Session L3 Multiphase Flows VI 325 - Ivan Zadrazil, Imperial College London

3:35PM L3.00001 Numerical and experimental study of disturbance wave development in vertical two-phase annular flow[1] , GEOFFREY HEWITT, JUNFENG YANG, YUJIE ZHAO, CHRISTOS MARKIDES, OMAR MATAR, Imperial College London — The annular flow regime is characterized by the presence of a thin, wavy liquid film driven along the wall by the shear stress exerted by the gas phase. Under certain liquid film Reynolds numbers, large disturbance waves are observed to traverse the interface, whose length is typically on the order of 20 mm and whose height is typically on the order of 5 times the thickness of the thin (substrate) layer between the waves. Experimental wok has been conducted to study the disturbance wave onset by probing the local film thickness for different Reynolds numbers. It is observed the disturbance waves grow gradually from wavy initiation and form the ring-like structure. To predict the wavy flow field observed in the experiment, 3D CFD simulations are performed using different low Reynolds number turbulence models and Large Eddy Simulation. Modeling results confirm that there is recirculation within the waves, and that they as a packet of turbulence traveling over a laminar substrate film. We also predict the coalescence and the break-up of waves leading to liquid droplet entrainment into the gas core.

[1]Skolkovo Foundation, UNIHEAT project

3:48PM L3.00002 The effect of surfactant on stratified and stratifying gas-liquid flows[1] , BAPTISTE HEILES, Ecole Normale Superieure de Cachan, IVAN ZADRAZIL, OMAR MATAR, Imperial College London — We consider the dynamics of a stratified/stratifying gas-liquid flow in horizontal tubes. This flow regime is characterised by the thin liquid films that drain under gravity along the pipe interior, forming a pool at the bottom of the tube, and the formation of large-amplitude waves at the gas-liquid interface. This regime is also accompanied by the detachment of droplets from the interface and their entrainment into the gas phase. We carry out an experimental study involving axial- and radial-view photography of the flow, in the presence and absence of surfactant. We show that the effect of surfactant is to reduce significantly the average diameter of the entrained droplets, through a tip-streaming mechanism. We also highlight the influence of surfactant on the characteristics of the interfacial waves, and the pressure gradient that drives the flow.

[1]EPSRC Programme Grant EP/K003976/1

4:01PM L3.00003 Quantitative consideration of flow structures (bubble swarms and liquid motion) and dissolved CO2 concentration transportation, in a bubbly flow , DAISUKE SHINOHARA, Graduate School of Engineering, Shizuoka University, TAKAYUKI SAITO, Research Institute of Green Science and Technology, Shizuoka University — The objective of the present study is to clarify the relationship between large scale flow structures (: bubble swarm and liquid motion) and dissolved CO2 concentration transportation, in a large-diameter bubble column. For this specific purpose, the time-series void fractions, dissolved CO2 concentration and liquid-phase-velocities were simultaneously measured by using a photoelectric optical fiber probe (POFP) and Laser Doppler Velocimetry. The POFP was newly developed in order to simultaneously measure bubble characteristics and dissolved CO2 concentration. We calculated the spatial scale of the bubble swarms and liquid motion based on the thinking of the integral length scale. The spatial scale of the bubble swarms and liquid motion was large in the bottom zone. Moreover, the size of this spatial scale changed with time; i.e. the flow structures changed with time in the bottom zone. The characteristics of the flow structures in the bottom zone faded out towards the upper zone of the column. The cross-correlation coefficients of dissolved CO2 concentration were calculated at several zones by height. As a result, the relationship between the flow structures and dissolved CO2 concentration transportation was found out.

4:14PM L3.00004 The Annular Two-phase Flow on Rod Bundle: The Effects of Spacers , TOMOAKI KUNUGI, SON PHAM, ZENSAKU KAWARA, TAKEHIKO YOKOMINE, Department of Nuclear Engineering, Kyoto University — The annular two-phase flow on rod bundle keeps an important role in many heat exchange systems but our knowledge about it, especially the interaction between the liquid film flowing on the rods' surfaces and the spacers is very limited. This study is aimed to the investigation of how the spacer affects the disturbance waves of the flow in a 3x3 simulating BWR fuel rod bundle test section. Firstly, the characteristics of the disturbance waves at both upstream and downstream locations of the spacer were obtained by using reflected light arrangement with a high speed camera Phantom V7.1 (Vision Research Inc.) and a Nikon macro lens 105mm f/2.8. The data showed that the parameters such as frequency and circumferential coherence of the disturbance waves are strongly modified when they go through the spacer. Then, the observations at the locations right before and after the spacer were performed by using the back light arrangement with the same high speed camera and a Cassegrain optical system (Seika Cooperation). The obtained images at micro-scale of time and space provided the descriptions of the wavy interface behaviors right before and after the spacer as well as different droplets creation processes caused by the presence of this spacer.

4:27PM L3.00005 Two-Phase Lattice Boltzmann Modeling of Boiling Phenomena , MAHMOOD MOHAMMADI SHAD, TAEHUN LEE, MASAHIRO KAWAJI, Mechanical Engineering Department, City College of City University of New York — Modern advanced technologies such as electronics cooling need large heat removal from surfaces. Nucleate boiling phenomena provides sufficient cooling for these purposes because of large value of latent heat stored in the liquid. A modified multiphase lattice Boltzmann equation model is developed for liquid-vapor phase change phenomena. The volumetric mass flow rate at the interface due to phase change is included in the non-zero value of divergence of velocity. The evolution equation for hydrodynamic pressure is used to force the incompressibility in the bulk regions and the compressibility in the interfacial region. The one-dimensional Stefan problem with analytical solution is used to validate the proposed model and the two-dimensional nucleate boiling on a flat surface is simulated as the main case study.

4:40PM L3.00006 Slug front gas entrainment in gas-liquid two-phase horizontal flow using hi-speed slug-tracking[1] , IVAN ZADRAZIL, OMAR MATAR, CHRISTOS MARKIDES, Imperial College London — A gas-liquid flow regime where liquid-continuous regions travel at high speeds (i.e. slugs) through a pipe separated by regions of stratified flow (i.e. elongated bubbles) is referred to as a "slug flow." This regime is characterised by the turbulent entrainment of gas into the slug front body. We use a high-speed camera mounted on a moving robotic linear rail to track the formation of naturally occurring slugs over 150 pipe diameters. We show that the dynamics of the slugs become progressively more complex with increasing liquid and gas Reynolds numbers. Based on the slug- tracking visualization we present, over a range of conditions: (i) phenomenological observations of the formation and development of slugs, and (ii) statistical data on the slug velocity and gas entrainment rate into the slug body.

[1]EPSRC Programme Grant EP/K003976/1

4:53PM L3.00007 Using DNS Data for Modeling of Bubbly Flows[1] , GRETAR TRYGGVASON, University of Notre Dame, JIACAI LU, Worcester Polytechnic Institute — Direct numerical simulations (DNS) of bubbly flows in vertical channels have lead to significant insight into the structure and dynamics of the flow. However, for applications to industrial systems the range of scales is sufficiently large so that DNS will remain impractical for the foreseeable future. Furthermore, there are indications that the dynamics at the smallest scales is sufficiently universal so that it should not be necessary to recomputed those scales to accurately predict the large-scale motion. Thus, models where the large-scale motion is computed and the unresolved scales are modeled will continue to be of importance. Here, we report two efforts to generate data that can be used to help modeling. In the first study we have computed the lift and drag force on a single bubble in shear flow under a variety of conditions, focusing on both the effect of the deformability of the bubble as well as the changes in drag and lift as the bubbles are placed close to a wall. In the other study we examine the transient dynamics of a large number of bubbles of different sizes, initially placed in a turbulent upflow in a vertical channel, and use the fully resolved DNS date to compute the various quantities that are generally unresolved in a model of the large-scale flow.

[1]Research supported by DOE (CASL)

5:06PM L3.00008 Two-phase viscous flows in channels with chemically patterned walls , VLADIMIR AJAEV, Southern Methodist University, ELIZAVETA GATAPOVA, OLEG A. KABOV, Institute of Thermophysics (Russia) — Recent experimental studies of two-phase channel flows past chemically patterned surface showed that the bubbles of gas phase in the liquid tend to accumulate in the regions of lower wettability. We investigate how the presence of such bubbles affects the flow, in particular the viscous resistance at a given imposed pressure gradient. Two regimes are considered. First, we study the limit of relatively low concentration of bubbles, obtaining expressions for effective slip past a hydrophobic stripe partially covered by bubbles. Then, we consider the regime when bubbles merge, leading to complete coverage of hydrophobic region by the gas phase. In the latter case, the viscous flow is affected by a competition between the slippage effect at the gas-liquid interface and the reduction of the channel flow cross-section. The shape of the gas-liquid interface is found from solving the coupled problems for the flows in two phases.

5:19PM L3.00009 A computational model for large eddy simulation of dilute bubbly turbulent flows[1] , MOHAMMAD HAJIT, FOTIS SOTIROPOULOS, St. Anthony Falls Laboratory, University of Minnesota — A mathematical formulation of filtered equations for two phase bubbly flows based on two-fluid method is presented. To remove high frequencies (noise), we extracted the filtered form of the equations in curvilinear coordinates, converting the microscopic governing equations to macroscopic equations via spatial averaging of solution variables. The set of equations describing the hydrodynamics in a gas-liquid system can be solved effectively if the solution procedure is decoupled so that an efficient iterative scheme can be employed. We propose a formulation for dilute bubbly flows in which the equations are converted to a loosely-coupled form. The resulting mathematical model is based on five distinct sets of equations, namely mixture momentum balance, pressure Poisson equation, Boyle's law and momentum and mass balances of gas phase. This mathematical formulation provides an efficient numerical procedure for two-way coupling of bubbly flows at low gas holdups. The subgrid-scale modeling is based on dynamic procedure of Germano for both phases. The formulation is validated for a fully turbulent bubble column test by comparing to available experimental results.

[1]This work is supported by the US department of energy (DE-EE0005416) and the Minnesota supercomputing institute.

5:32PM L3.00010 Numerical simulation of cavitating channel flows including non-condensable gases effects , MICHELE BATTISTONI[1], SIBENDU SOM, DOUGLAS E. LONGMAN, Argonne National Laboratory, Chicago IL — Fuel injectors often feature cavitation because of large pressure gradients which in some regions lead to extremely low pressure levels. Numerical results are assessed against quantitative high resolution experimental data collected at Argonne National Laboratory using synchrotron x-ray radiography on real-size fuel nozzles. Simulation are performed on structured embedded grids using finite volume method and second-order discretization schemes in space and time. A single fluid homogeneous mixture model is compared to a multi-fluid non-homogeneous model. Two mass transfer models for predicting cavitation are also studied. RANS and LES cases are presented. The presence of dissolved gases in the multi-phase flow is addressed and their effect has been accounted for by running compressible three-phase flow simulations. The study highlights the importance of accounting for dissolved gases in the liquid, since some void formations, which could be attributed to cavitation, are actually due to non-condensable gas expansion. A discussion about the effect of turbulent pressure fluctuations on cavitation inception is also presented.

[1]Visiting Scholar at Argonne National Laboratory, Chicago IL; Assistant Professor at University of Perugia, Italy

5:45PM L3.00011 Wall drag modification by large droplets in turbulent channel flow , LUCA SCARBOLO, ALFREDO SOLDATI, University of Udine — Object of this work is to examine the influence of large deformable droplets on wall-bounded turbulence. To this aim we study the behavior of a swarm of droplets with the same density and viscosity of the surrounding fluid in turbulent channel flow. We use direct numerical simulations of turbulence coupled with a phase field model for the interface tracking. A wide range of Weber numbers (ratio between inertia and surface tension) is explored for shear Reynolds number $Re_\tau = 150$. To quantify surface tension effects on the flow the wall shear stress, the average droplet deformability and the turbulent kinetic energy budgets will be analyzed.

Monday, November 25, 2013 3:35PM - 5:45PM –

Session L4 General Fluid Dynamics I: Drag Reduction 326 - Mitul Luhar, California Institute of Technology

3:35PM L4.00001 Drag reduction due to spatial thermal modulations , JERZY M. FLORYAN, University of Western Ontario, DANIEL FLORYAN, Cornell University — It is demonstrated that a significant drag reduction for pressure driven flows can be realized by applying spatially distributed heating. The heating creates separation bubbles that separate the stream from the bounding walls and, at the same time, alters distribution of the Reynolds stress providing a propulsive force. The strength of this effect is of practical interest for heating with the wave numbers 0(1) and for flows with small Reynolds numbers and, thus, it is of potential interest for applications in micro-channels. The strength of the effects can be increased by using heating with a non-zero mean. The drag reducing effect increases proportionally to the second power of the heating intensity. This increase saturates if the heating becomes too intense.

3:48PM L4.00002 Modeling drag reduction by slippery surfaces comprised of microridges with two fluids[1] , MOHAMED A. SAMAHA, MARCUS HULTMARK, Princeton University — Theoretical analysis, numerical simulations, and experimental study are developed to predict drag reduction possessed by slippery surfaces, which entrap a second immiscible fluid within their micropores. The aim is to improve our understanding of the slip-flow and drag-reduction effects in terms of surface morphology and properties of both fluids. Stokes flow is simulated over surfaces with microstructure of both streamwise and spanwise ridges configurations. The entrapped fluid circulation between the microridges is also simulated for different geometries of the cavity. For validation, the results of the theoretical model are compared to those of the numerical simulations, and also compared to the available results of previous studies reported in the literature. Scaling laws are obtained for the reduction in shear stress at the surface in terms of the generic surface characteristics (surface roughness and both fluids' properties). These predictions are compared to the experimental data of rheological tests performed on fabricated samples. The models allow using different kinds of fluids with a wide range of viscosities. This work could be utilized to design slippery surfaces such as superhydrophobic and omniphobic coatings to maximize drag reduction.

[1]Office of Naval Research (Grant #: N00014-12-1-0875), program manager Dr. Ki-Han Kim.

4:01PM L4.00003 Groove Optimization for Drag Reduction , A. MOHAMMADI, JERZY FLORYAN, University of Western Ontario — It has been shown that long-wavelength, longitudinal grooves reduce pressure losses in laminar, pressure driven flows. This work is focused on the search for the groove shapes that maximize the reduction of such losses. It is shown that the optimal shapes can be characterized using reduced geometry models involving just a few Fourier modes. Two classes of grooves have been considered, i.e. equal-depth grooves, which have the same height and depth, and unequal-depth grooves. It has been shown that the optimal grooves in the former cases are characterized by a certain universal trapezoid. There exists an optimum depth in the latter case and this depth, combined with the corresponding groove shape, defines the optimal geometry; this shape is well-approximated by a delta function. The maximum possible drag reduction has been determined for the optimal shapes. The analysis has been extended to kinematically-driven flows. It has been shown that in this case the longitudinal grooves always increase flow resistance regardless of their shape.

4:14PM L4.00004 Drag reduction using slippery liquid infused surfaces[1] , MARCUS HULTMARK, HOWARD STONE, ALEXANDER SMITS, IAN JACOBI, MOHAMED SAMAHA, JASON WEXLER, JESSICA SHANG, BRIAN ROSENBERG, LEO HELLSTRÖM, YUYANG FAN, Princeton University — A new method for passive drag reduction is introduced. A surface treatment inspired by the Nepenthes pitcher plant, previously developed by Wong et al. (2011), is utilized and its design parameters are studied for increased drag reduction and durability. Nano- and micro-structured surfaces infused with a lubricant allow for mobility within the lubricant itself when the surface is exposed to flow. The mobility causes slip at the fluid-fluid interface, which drastically reduces the viscous friction. These new surfaces are fundamentally different from the more conventional superhydrophobic surfaces previously used in drag reduction studies, which rely on a gas-liquid interface. The main advantage of the liquid infused surfaces over the conventional surfaces is that the lubricant adheres more strongly to the surface, decreasing the risk of failure when exposed to turbulence and other high-shear flows. We have shown that these surfaces can reduce viscous drag up to 20% in both Taylor-Couette flow and in a parallel plate rheometer.

[1]Supported under ONR Grants N00014-12-1-0875 and N00014-12-1-0962 (program manager Ki-Han Kim)

4:27PM L4.00005 Drag reduction using a multi-cavity at the afterbody[1] , ENRIQUE SANMIGUEL-ROJAS, Universidad de Córdoba, Spain, ANTONIO MARTÍN-ALCÁNTARA, CÁNDIDO GUTIÉRREZ-MONTES, CARLOS MARTÍNEZ-BAZÁN, Universidad de Jaén, Spain, MANUEL A. BURGOS, Universidad Politécnica de Cartagena, Spain, MANUEL HIDALGO-MARTÍNEZ, Universidad de Córdoba, Spain — We present a numerical study on the drag reduction of a two-dimensional bluff body with a blunt trailing edge, which has a chord length L, body height H and spanwise width W, being $H/W \ll 1$, aligned with a turbulent incompressible free-stream of velocity U_∞, density ρ and viscosity μ. In particular, an extensive parametric study is performed numerically using the IDDES turbulent model, at a Reynolds number, $Re = \rho U_\infty H/\mu = 20000$, to analyze the effect on the drag coefficient C_D of both a single-cavity as a multi-cavity of variable depth h at the base of the body. It is observed within the range, $0 \leq h/H \leq 0.2$, that C_D decreases monotonically reaching an asymptotic value in both cases. In turn, shorter cavity depths are necessary to reach the same drag reduction with a multi-cavity than with a single-cavity. On the other hand, the temporal evolution of the drag coefficient shows a lower standard deviation with a multi-cavity than with a single-cavity, which is manifested in the flow as a wake with a lower level of disorder.

[1]This work was supported by Junta de Andalucía under project PI10-TEP5702

4:40PM L4.00006 Longevity and drag reduction of omniphobic surfaces[1] , BRIAN ROSENBERG, MOHAMED A. SAMAHA, IAN JACOBI, JESSICA SHANG, MARCUS HULTMARK, Princeton University, ALEXANDER SMITS, Princeton University, Monash University — Omniphobic surfaces, which consist of an omniphobic lubricant impregnated into a micro/nanoscale textured substrate, have been shown to repel a wide range of liquids [Wong et. al (Nature 2011)]. Here, experiments are performed on these surfaces to investigate the drag reduction as well as the time-dependent omniphobicity in the presence of flow. Drag measurements are performed in number of different flows including parallel plate and Taylor-Couette rheometers, pipe flow, and bluff body flows. The longevity of the surfaces are measured using three techniques: (i) an in situ noninvasive optical method to characterize the the loss of lubricant with time; (ii) thin-film interferometry measurements of the lubricant thickness versus time; and (iii) goniometer measurements of the time-dependent threshold sliding angle as well as contact-angle hysteresis. The impact of the substrate morphology on the drag reduction and longevity is observed both with and without flow in the surrounding water environment. This work could help to investigate ways of enhancing the drag-reducing properties of omniphobic surfaces by controlling their morphologies.

[1]Supported under ONR Grants N00014-12-1-0875 and N00014-12-1-0962 (program manager Ki-Han Kim)

4:53PM L4.00007 Drag Reduction On Multiscale Superhydrophobic Surfaces[1] , ELLIOT JENNER, University of Pittsburgh, CHARLOTTE BARBIER, Oakridge National Laboratory, BRIAN D'URSO, University of Pittsburgh — Fluid drag reduction is of great interest in a variety of fields, including hull engineering, microfluidics, and drug delivery. We fabricated samples with multi-scale superhydrophobic surfaces, which consist of hexagonally self-ordered microscopic spikes grown via anodization on macroscopic grooves cut in aluminum. The hydrodynamic drag properties were studied with a cone-and-plate rheometer, showing significant drag reduction near 15% in turbulent flow and near 30% in laminar flow. In addition to these experiments, numerical simulations were performed in order to estimate the slip length at high speeds. Furthermore, we will report on the progress of experiments with a new type of surface combining superhydrophobic surfaces like those discussed above with Slippery Liquid Infused Porous Surfaces (SLIPS), which utilize an oil layer to create a hydrophobic self-repairing surface. These "Super-SLIPS" may combine the best properties of both superhydrophobic surfaces and SLIPS, by combining a drag reducing air-layer and an oil layer which may improve durability and biofouling resistance.

[1]This research was supported by the ORNL Seed Money Program. Oak Ridge National Laboratory is managed by UT-Battelle, LLC, for the U.S. Dept. of Energy under contract DE-AC05-00OR22725.

5:06PM L4.00008 Numerical investigation of drag in regular arrays of circular cylinders , SATOSHI YOKOJIMA, Department of Mathematical and Systems Engineering, Shizuoka University, YOSHIHISA KAWAHARA, Department of Civil and Environmental Engineering, Hiroshima University — The temporal and spatial distribution of the drag exerted by regular arrays of circular cylinders is closely investigated by a series of numerical simulations. Applicability of the macroscopic drag-force model for the flow is also examined.

5:19PM L4.00009 Effective Medium Theory for Drag Reducing Micro-patterned Surfaces in Turbulent Flows , ILENIA BATTIATO, Clemson University — Inspired by the lotus effect, many studies in the last decade have focused on micro- and nano-patterned surfaces. They revealed that patterns at the micro-scale combined with high contact angles can significantly reduce skin drag. However, the mechanisms and parameters that control drag reduction, e.g. Reynolds number and pattern geometry, are still unclear. We propose an effective medium representation of the micro-features that treats the latter as a porous medium, and provides a framework to model flows over patterned surfaces in both Cassie and Wenzel states. Our key result is a closed-form expression for the skin friction coefficient in terms of frictional Reynolds (or Karman) number in turbulent regime. We apply the proposed model to turbulent flow over superhydrophobic ridged surfaces. The model predictions agree with laboratory experiments for Reynolds numbers ranging from 3000 to 10000.

5:32PM L4.00010 Fly in Atmosphere by Drag Force – Easy Thrust Generation Aircraft Engine Based Physics , MWIZERWA PIERRE CELESTIN, None — This paper aims to present to the science community another way to fly in atmosphere, a way which is much more cheaper, efficient, safe and easy. Over the years scientists have been trying to find a way to built the vertically taking off vehicles but there have been no satisfactory success(what have been found was very expensive), Even aircrafts we know now need very sophisticated and expensive engines and not efficient enough. This way of flying may help our governments to spend less money on technologies and will help people to travel at very low prices so that, it may be a solution to the crisis which the world faces nowadays. In other words, it is my proposal to the next generation technologies we was looking for for years because everything can fly from the car to the trucks, the spaceships and even the hotels maybe constructed and fly as we construct the ships which sail in the oceans. My way of flying will have many applications in all the aspect of travel as it is going to be explained.

Monday, November 25, 2013 3:35PM - 5:19PM –

Session L5 CFD VI 327 - Meng-Sing Liou, NASA

3:35PM L5.00001 On Numerical Heating[1] , MENG-SING LIOU, NASA Glenn Research Center — The development of computational fluid dynamics over the last few decades has yielded enormous successes and capabilities that are being routinely employed today; however there remain some open problems to be properly resolved. One example is the so-called overheating problem, which can arise in two very different scenarios, from either colliding or receding streams. Common in both is a localized, numerically over-predicted temperature. Von Neumann reported the former, a compressive overheating, nearly 70 years ago and numerically smeared the temperature peak by introducing artificial diffusion. However, the latter is unphysical in an expansive (rarefying) situation; it still dogs every method known to the author. We will present a study aiming at resolving this overheating problem and we find that: (1) the entropy increase is one-to-one linked to the increase in the temperature rise and (2) the overheating is inevitable in the current computational fluid dynamics framework in practice. Finally we will show a simple hybrid method that fundamentally cures the overheating problem in a rarefying flow, but also retains the property of accurate shock capturing. Moreover, this remedy (enhancement of current numerical methods) can be included easily in the present Eulerian codes.

[1]This work is performed under NASA's Fundamental Aeronautics Program.

3:48PM L5.00002 Numerical Investigation of Conjugate Heat Transfer in a Channel with a Growing Deposit Layer , HONGYING LI, IHPC A*STAR, YITFATT YAP, The Petroleum Institute, Abu Dhabi, United Arab Emirates, JING LOU, IHPC A*STAR, JOHN CHAI, The Petroleum Institute, Abu Dhabi, United Arab Emirates — The working fluid carries particles flowing in channels is widely encountered in many engineering applications such as oil and gas pipes and heat exchangers. These particles have a tendency to deposit onto the wall of the channels, form a deposit layer. This additional growing and increasingly thicker deposit layer, normally is of a lower thermal conductivity. In the system with heat transfer involved, such deposit layer introduces extra thermal resistance and consequently leads to the lower the heat transfer performance of the system. Besides, the deposit layer reduces flow cross sectional area of the channel and directly responsible for inducing a larger pressure drop. As such, a good understanding of the conjugate heat transfer coupling the evolving deposit layer and fluid flow is important. This numerical study is undertaken to fill in some of the gaps in this respect. Here, we consider conjugate heat transfer in a channel with a deposit layer gradually growing on the wall. The problem is governed by conservation equations for mass, momentum, species and energy, coupled with the appropriate interfacial condition at the depositing front separating the fluid from the deposit. This is a moving boundary problem as the front evolves over time. The depositing front is captured using the level-set method in this study. Numerical solution is performed on a fixed mesh using the finite volume method. A detailed parametric study quantifying the effect of the growing deposit layer on the heat transfer performance is performed.

4:01PM L5.00003 Upscale and downscale energy transfer in turbulent open channel flow , SALVATORE LOVECCHIO, Dipartimento di Ingegneria Elettrica, Gestionale e Meccanica, University of Udine, ALFREDO SOLDATI, Centro Interdipartimentale di Fluidodinamica e Idraulica and Dipartimento di Energetica e Macchine, University of Udine — Heat and mass transfer phenomena in free-surface turbulence are of great importance in a wide range of geophysical/environmental situations. Examples include CO_2 transfer across the ocean surface or the transport of organic species. These phenomena are controlled by the dynamics of free-surface turbulent structures, which are known to give rise to transport of energy among the flow scales. In this study we use Direct Numerical Simulation to analyze such energy transfer in turbulent channel flow with a free surface. Our results suggest that the inhomogeneity, inherently present in near-wall and free-surface turbulence, generates energy fluxes that correspond to a spatial redistribution of turbulent kinetic energy within the flow. We show that the energy transfer near the boundaries is significantly different from that in the bulk flow, where the behaviour is more homogeneous and isotropic. This is due to an increased energy backscatter from small to large flow scales. We also show that regions of direct (downscale) and inverse (upscale) energy transfer can be associated to the coherent structures of the flow.

4:14PM L5.00004 Experimental Validation Dataset for CFD Simulations of Buoyancy Opposed Convection[1] , JEFF HARRIS, BLAKE LANCE, BARTON SMITH, Utah State University — New experiments in the Rotatable Buoyancy Tunnel are described. This unique facility was built specifically for computational fluid dynamics (CFD) validation experiments in natural, forced, and mixed convection in both buoyancy aided or opposed scenarios. The tunnel features clear walls for non-intrusive optical measurements, a heated wall (controlled to isothermal or constant flux conditions), and the capability to invert without changing the inlet or as-built dimensions. The wall temperature and inlet temperature are measured, along with the inlet velocity and turbulence profiles, to define simulation boundary conditions. The experiment includes acquisition of particle image velocimetry data at several streamwise locations in the boundary layer along the heated plate. Heat flux at those locations is also measured. The flow consists of natural convection driving air upwards combined with forced convection (fan driven) drawing air down. A RANS CFD simulation for this scenario is presented, with a comparison of several models' computed boundary layer flow, heat flux, and pressure drop to the measured values of the same.

[1]This research is being performed using funding received from the DOE Office of Nuclear Energy's Nuclear Energy University Programs.

4:27PM L5.00005 Computational Fluid Dynamics Uncertainty Analysis applied to Heat Transfer over a Flat Plate , CURTIS GROVES, University of Central Florida and NASA Kennedy Space Center, MARCEL ILIE, University of Central Florida, PAUL SCHALLHORN, NASA Kenndy Space Center — There have been few discussions on using Computational Fluid Dynamics (CFD) without experimental validation. Pairing experimental data, uncertainty analysis, and analytical predictions provides a comprehensive approach to verification and is the current state of the art. With pressed budgets, collecting experimental data is rare or non-existent. This paper investigates and proposes a method to perform CFD uncertainty analysis only from computational data. The method uses current CFD uncertainty techniques coupled with the Student-T distribution to predict the heat transfer coefficient over a flat plate. The inputs to the CFD model are varied from a specified tolerance or bias error and the difference in the results are used to estimate the uncertainty. The variation in each input is ranked from least to greatest to determine the order of importance. The results are compared to heat transfer correlations and conclusions drawn about the feasibility of using CFD without experimental data. The results provide a tactic to analytically estimate the uncertainty in a CFD model when experimental data is unavailable.

4:40PM L5.00006 ABSTRACT WITHDRAWN –

4:53PM L5.00007 Fast geometric sensitivity analysis in hemodynamic simulations using a machine learning approach , SETHURAMAN SANKARAN, Senior Computational Scientist, LEO GRADY, Vice president, Research and Development, CHARLES TAYLOR, CTO — In the cardiovascular system, blood flow rate, velocities and blood pressure are governed by the Navier-Stokes equations. Inputs to the system such as (a) geometry of arterial tree, (b) clinically measured blood pressure and viscosity, (c) boundary resistances, among others, are typically uncertain. Due to a large number of such parameters, there is a need to efficiently quantify uncertainty in solution fields in this multi-parameter space. We use a machine learning approach to approximate the simulation-based solution. Using an offline database of pre-computed solutions, we compute a map (rule) from the features to solution fields. This is coupled to an adaptive stochastic collocation method to quantify uncertainties in input parameters. We achieve significant speed-up ($\sim$1000 fold) by approximating the simulation-based solution using a machine learning predictor. Bagged decision tree was found to be the best predictor among many candidate regressors (correlation coefficient $\sim$0.92). The sensitivities obtained using machine learning approach has a correlation coefficient of 0.91 with those obtained using finite element simulations. We also calculated and ranked the impact of different inputs such as problem geometry, and clinical parameters. We observed that the impact of geometry supersedes the impact of other variables. Mostly, segments with significant disease in the larger arteries had the highest sensitivities. We were able to localize sensitive regions in long segments with a focal disease using a multi-resolution approach.

5:06PM L5.00008 ABSTRACT WITHDRAWN –

Monday, November 25, 2013 3:35PM - 5:45PM –

Session L6 Microfluids: Fluidic Devices II 328 - Manu Prakash, Stanford University

3:35PM L6.00001 Mosquitoes meet microfluidics: High-throughput microfluidic tools for insect-parasite ecology in field conditions[1] , MANU PRAKASH, HARIPRIYA MUKUNDARAJAN, Stanford University — A simple bite from an insect is the transmission mechanism for many deadly diseases worldwide — including malaria, yellow fever, west nile and dengue. Very little is known about how populations of numerous insect species and disease-causing parasites interact in their natural habitats due to a lack of measurement techniques. At present, vector surveillance techniques involve manual capture by using humans as live bait, which is hard to justify on ethical grounds. Individual mosquitoes are manually dissected to isolate salivary glands to detect sporozites. With typical vector infection rates being very low even in endemic areas, it is almost impossible to get an accurate picture of disease distribution, in both space and time. Here we present novel high-throughput microfluidic tools for vector surveillance, specifically mosquitoes. A two-dimensional high density array with baits provide an integrated platform for multiplex PCR for detection of both vector and parasite species. Combining techniques from engineering and field ecology, methods and tools developed here will enable high-throughput measurement of infection rates for a number of diseases in mosquito populations in field conditions.

[1]Pew Foundation

3:48PM L6.00002 Scale reduction impact on bacterial growth , DAVID LALANNE-AULET, PIERRE GUILLOT, Rhodia/Solvay-Laboratory of the Future, ANNIE COLIN, University of Bordeaux I, PHILIPPE MARCHAL, Solvay - RICL — Miniaturized tools for microbiological tests intensification have proven their impressive potential among the past decade and keep focusing a lot of researches. However, systematic comparison with usual tests is still lacking and prevents thus the implementation of these new methods. In this work, we study the scale effects on the growth of a bacterial population in order to identify growth-limiting parameters and determine ranges in which miniaturized tools really mimic usual tests. Incubations are performed in miniaturized droplets engineered in microfluidic devices with biocompatible fluorinated oil. This oil phase replaces the usual atmosphere as gas reservoir. The impact of size and environment modifications on microbial growth has to be evaluated. At first sight, system's size reduction is favorable because it minimizes nutrients diffusion times. However, the amount of needed gas available (O2) and the ability of storing undesirable gas (CO2) become limited. We show that oxygen does not limit the growth, whereas carbon dioxide accumulation can decrease growth yields by various mechanisms that will be discussed. Through this study we optimize growth conditions in miniaturized tools for long-term cultures.

4:01PM L6.00003 Simulation of actuated synthetic cilia expelling microorganisms from a surface , HENRY SHUM, ANURAG TRIPATHI, University of Pittsburgh, JULIA YEOMANS, University of Oxford, ANNA BALAZS, University of Pittsburgh — The continual motion of cilia provides a defense against biofouling for a variety of marine organisms. Inspired by this natural solution, we perform numerical simulations to study the interactions between actuated, biomimetic cilia and model microswimmers that are hydrodynamically attracted to bare surfaces and therefore behave similarly to fouling organisms. The 3-dimensional fluid flow is coupled to the motion of the cilia and swimmers using an integrated lattice Boltzmann and immersed boundary method. We show that a sparse array of actuated cilia, through which the organisms are able to swim, is able to robustly expel swimmers. The average residence time of the swimmers in the ciliary layer is reduced if the motion of the cilia generates a net fluid flow, but for relatively fast swimmers, the steric interaction with the moving cilia alone is sufficient to "knock" the organisms away from the surface. As the demonstrated mechanisms for antifouling do not rely on specific chemical or physical properties of the surface or swimmer, actuated cilia can potentially protect microfluidic devices, filtration membranes or ship hulls from a wide range of fouling species.

4:14PM L6.00004 Dancing Droplets , NATE CIRA, MANU PRAKASH, Stanford University — Inspired by the observation of intricate and beautifully dynamic patterns generated by food coloring on corona treated glass slides, we have investigated the behavior of propylene glycol and water droplets on clean glass surfaces. These droplets exhibit a range of interesting behaviors including long distance attraction or repulsion, and chasing/fleeing upon contact. We present explanations for each of these behaviors, and propose a detailed model for the long distance interactions based on vapor facilitated coupling. Finally we use our understanding to create several novel devices which: passively sort droplets by surface tension, spontaneously align droplets, drive droplets in circles, and cause droplets to bounce on a vertical surface. The simplicity of this system lends it particularly well to application as a toy model for physical systems with force fields and biological systems such as chemotaxis and motility.

4:27PM L6.00005 Synchronous Droplet Microfluidics: One "Clock" to rule them all , GEORGIOS KATSIKIS, MANU PRAKASH, Stanford University — Controlling fluid droplets efficiently in the microscale is of great interest both from a basic science and a technology perspective. Here we demonstrate a general-purpose, highly scalable microfluidic control strategy through a single global clock signal that enables synchronous control of arbitrary number of droplets in a planar geometry. A rotating precessive magnetic field provides the clock signal, enabling simultaneous control of droplet position, velocity and trajectories. Using high-speed video capture and computational droplet tracking, we characterize a number of propagation circuits. Successful propagation depends on driving frequency and the size of the droplets, which is characterized as a regime diagram and rationalized in terms of Stokes and Capillary numbers. Novel interaction regimes for hydrodynamic interaction between droplets are also identified, paving the way for building complex synchronous fluidic circuits in the future.

4:40PM L6.00006 Estimation of manipulation force for droplet in O/W system under photothermal interfacial control , MASAHIRO MOTOSUKE, MASAKAZU MUTO, Tokyo University of Science — Droplet-based microfluidics has been keenly investigated as a discrete operation of tiny amount of reagent or individual cell inside droplets. Noncontact maniplulation of droplets in a microfluidic platform can be achieved utilizing a photothermally induced interfacial tension gradient. Although this method could provide flexible and selective toolbox for droplet control using patterned light irradiation instead of complexed channel geometry. In this study, an experimental estimation of the manipulation force for droplet under photothermal interfacial control is presented. Temperature distribution in a PDMS microfluidic device was quantified by laser-induced fluorescence based on thermal quenching of fluorecein. Under trapped condition in a steady flow, the exerted force was determined considering a balance between the drag and the interfacial force. The results indicate that the nN-order force is available in the photothermal interfacial activation and imply the applicability of this method for a versatile droplet-based microfluidic platform.

4:53PM L6.00007 ABSTRACT WITHDRAWN —

5:06PM L6.00008 Two-phase droplet injectors for studies at X-ray free-electron laser facilities , CLAUDIU STAN, SLAC National Accelerator Facility — Hard X-ray free-electron lasers (XFEL) such as the recently developed Linac Coherent Light Source (LCLS) at SLAC deliver pulses with extremely short duration and intensities many orders of magnitude larger than previous sources, to enable visualization of the motion of single atoms within condensed matter. To circumvent X-ray damage, imaging experiments at LCLS are performed serially, with new samples being brought to the vacuum interaction region with X-ray pulses. Continuous liquid microjets in vacuum are currently the best way of carrying and regenerating the samples, but they consume inefficiently scarce samples, such as membrane protein microcrystals. To solve this problem, and to enhance the accuracy of pump-probe experiments, we are developing a two-liquid delivery method in which the sample is carried in disperse-phase drops contained in an immiscible continuous-phase liquid. We will report on (i) the phase-locked generation of sample-carrying droplets with an electrically-assisted axisymmetric flow-focusing device, (ii) methods to reduce the accumulation of phase jitter in the timing of drops during transport, and we will discuss methods for (iii) separating the continuous and disperse phases and (iv) ejection of sample-containing drops into air or vacuum.

5:19PM L6.00009 Ink-jet patterned superhydrophobic paper for open-air surface microfluidic devices , MOHAMED ELSHARKAWY, THOMAS SCHUTZIUS, CONSTANTINE MEGARIDIS, University of Illinois at Chicago — We present the production of superhydrophobic paper via polymer solution drop-casting on silicon carbide paper. The resulting substrate is patterned using household inkjet printers. The patterning process yields the ability to produce regions of varying wettability by simply controlling the intensity of ink deposited and surface area over which the ink is applied. By manipulating the two previously mentioned parameters we can develop surfaces that are capable of selective droplet sliding and adhesion. The mentioned methodology has produced superhydrophobic paper of advancing angles $157° \pm 4.5°$, receding angles $130° \pm 6.3°$, and droplet sliding angles of $13° \pm 2.3°$. We demonstrate the ability to vary the sliding angles of 10μL water droplets from $13°$ to $40°$ by printing lines of a constant intensity but varied width (.1 mm to 2 mm). It is thus possible to produce open-air surface microfluidic devices that are capable of pumpless transport, mixing, and rapid droplet sampling. The ease of the patterning technique allows for any imaginable 2D device to be printed, restricted only by the pattern usability and functionality. Lastly, post processing of printed areas using pH indicator solutions has demonstrated the use of these substrates in the area of Point-of-Care diagnostics.

5:32PM L6.00010 Design of Micropost Array for Low Bubble Retention[1] , MAHSHID MOHAMMADI, KENDRA SHARP, Oregon State University — One of the well-known problems in microfluidic systems is the presence of immobile bubbles which may disturb the performance of the device. Bubbles moving through variable cross sectional areas such as regions between microposts are prone to getting stuck in the contractions. The minimal external pressure needed to overcome the capillary pressure and drive a bubble out of a contraction is called the clogging pressure. At low flow rates the clogging pressure for a bubble moving through contractions between microposts may be much larger than the pressure drop along the length of the bubble, and the chances of bubble stagnation are high. Large bubbles which come into contact with several microposts have the highest potential for geometry-based management. With an appropriate design it is possible to restrict a large bubble between two adjacent columns of microposts and force it to elongate along the direction of the flow. In that situation the bubble experiences a larger pressure drop along its length and is more likely to overcome the resistant capillary pressure. Preventing the bubbles from taking meandering paths is a key factor for keeping bubbles in motion toward the desired destination. Based on our experimental evidence we propose a design criterion that facilitates bubble mitigation in a micropost arrangement. The criterion places geometrical constraints on longitudinal, transverse, and diagonal pitches that need to be satisfied in order to have low bubble retention characteristics in a micropost array.

[1]This work has been supported by National Institutes of Health contract number 1 R01 EB011567-01A1.

Monday, November 25, 2013 3:35PM - 5:45PM –

Session L7 Microfluids: Particles III - Droplets and Emulsions 329 - Shahab Shojaei Zadeh, Rutgers University

3:35PM L7.00001 Lateral migration of a droplet by optical force in a uniform flow , HYUNJUN CHO, JIN HO JUNG, CHEONG BONG CHANG, HYUNG JIN SUNG, KAIST — The lateral migration of a droplet under an optical force in a uniform flow was studied to show the separation characteristics in the cross-type optical separator. The initially spherical droplet was moved through the fluid flow using a loosely-focused Gaussian laser beam along the direction normal to the flow. To simulate such a system, the lattice Boltzmann method was adopted to obtain the two-phase flow field, and the dynamic ray tracing method was applied to the optical force calculation. The optical forces acting on the spherical droplets are found to be in good agreement with the theoretically predicted values. The trajectories of the droplets are obtained and compared with the experimental data. Simulations on various flow and optical parameters have been made. It is found that the lattice Boltzmann method, combined with the dynamic ray tracing, can be utilized to design optical manipulation systems.

3:48PM L7.00002 Effects of surfactants on the deformation of microfluidic drops[1] , MARIA LUISA CORDERO, Departamento de Fisica, FCFM, Universidad de Chile, CAMILO ULLOA, De — A microfluidic analog of the four-roll-mill experiment is used to study the deformation and breakup of microfluidic drops. The behavior of water drops flowing in mineral oil is quantified as a function of the capillary number, Ca, which is based on the oil viscosity, drop radius, flow shear rate and equilibrium interfacial tension, both in the presence and absence of surfactants. In the absence of surfactants the deformation of the drops increases linearly with Ca. If surfactants are added to the carrier oil then, for the same value of Ca, drops deform less if the flow velocity is larger. Moreover, for a given drop size in the presence of surfactants, drops begin to split at a threshold shear rate but stop breaking if the shear rate is increased beyond a second threshold. These observations are explained by a decrease in the surfactant concentration at the surface of the drop due to advection of surfactant molecules by the oil flow. This increases the interfacial tension, thus making the drop less deformable for higher flow velocities. We use the deformation of the drops to infer the mean interfacial tension and from this we quantify the surface concentration of surfactants at the drop interface.

[1]Work supported by FONDECYT 11100204

4:01PM L7.00003 Droplet velocity in microfluidics Hele-Shaw cell: effect of the disjoining pressure , AXEL HUERRE, MMN, UMR CNRS 7083, ESPCI ParisTech, 75005 Paris, France, OLIVIER THEODOLY, LAI, INSERM U600, CNRS UMR 6212, Case 937, 13009 Marseille, France, ISABELLE CANTAT, IPR, UMR CNRS 6251, Universite de Rennes 1, 35000 Rennes, France, ALEXANDER LESHANSKY, Department of Chemical Engineering, Technion-IIT, Haifa, 32000, Israel, MARIE-PIERRE VALIGNAT, LAI, INSERM U600, CNRS UMR 6212, Case 937, 13009 Marseille, France, MARIE-CAROLINE JULLIEN, MMN, UMR CNRS 7083, ESPCI ParisTech, 75005 Paris, France — We present an experimental evidence of the disjoining pressure effect on the traveling velocity of highly confined viscous droplets in microfluidics. Two regimes are observed depending on the capillary number, $Ca = \mu_f U_d/\gamma$. Above a critical capillary number Ca^* the droplet velocity U_d follows $U_d \propto U_f Ca^{1/3}$, where U_f is the velocity of the carrier liquid. However, for $Ca < Ca^*$, U_d does no longer depend on Ca. We present a direct *in-situ* measurement of the lubricating film thickness. For the capillary dependent regime, we recover the classical Bretherton's scaling: $h_\infty \propto e\,Ca^{2/3}$, where e is the cell thickness. However, for $Ca < Ca^*$ the film thickness is constant and set by the disjoining pressure. For $Ca \sim Ca^*$ we observe a dynamic coexistence of two film thicknesses, a signature of an oscillatory disjoining pressure. Based on the previous work and using scaling arguments, we propose a model that is able to reproduce the experimental results.

4:14PM L7.00004 Formation and dynamics of partially non-wetting droplets in square microchannels[1] , BIBIN M. JOSE, THOMAS CUBAUD, Stony Brook University — We experimentally study the formation and evolution of partially non-wetting droplets in microchannels made of glass and silicon. Droplets are generated by focusing pure water in an external phase of silicone oil using square microchannels. To probe the influence of the capillary number on droplet behavior in confined geometries, the oil viscosity is varied over four decades. For each fluid pair, the critical speed associated with the dynamic wetting transition is experimentally determined using a contact angle goniometer equipped with a high-speed camera. We discuss a variety of wetting phenomena in microchannels. In particular, we focus on the droplet lubrication transition from low to large capillary numbers and we examine droplet velocity as a function of droplet length, flow rates, and dynamic wetting properties.

[1]This work is supported by NSF (CBET- 1150389)

4:27PM L7.00005 Controlled Gelation of Particle Suspensions Using Controlled Solvent Removal in Picoliter Droplets , SHARON VUONG, LYNN WALKER, Department of Chemical Engineering, Carnegie Mellon University, SHELLEY ANNA, Department of Chemical Engineering and Department of Mechanical Engineering, Carnegie Mellon University — Droplets in microfluidic devices have proven useful as uniform picoliter reactors for nanoparticle synthesis and as components in tunable emulsions. However, there can be significant transport between the component phases depending on solubility and other factors. In the present talk, we show that water droplets trapped within a microfluidic device for tens of hours slowly dehydrate, concentrating the contents encapsulated within. We use this slow dehydration along with control of the initial droplet composition to monitor gelation of aqueous suspensions of spherical silica particles (Ludox) and disk-shaped clay particles (Laponite). Droplets are generated in a microfluidic device containing small wells that trap the droplets. We monitor the concentration process through size and shape changes of these droplets as a function of time in tens of droplets and use the large number of individual reactors to generate statistics regarding the gelation process. We also examine changes in suspension viscosity through fluorescent particle tracking as a function of dehydration rate, initial suspension concentration and initial droplet volume, and added salt, and compare the results with the Krieger-Dougherty model in which viscosity increases dramatically with particle volume fraction.

4:40PM L7.00006 Geometrically-protected reversibility in hydrodynamic Loschmidt-echo experiments , RAPHAËL JEANNERET, ESPCI-Université Paris Diderot, DENIS BARTOLO, ESPCI-ENS Lyon, BARTOLO LAB TEAM — We demonstrate an archetypal Loschmidt-echo experiment where thousands of droplets interact in a reversible fashion via a viscous fluid. Firstly, we show that, unlike equilibrium systems, periodically driven microfluidic emulsions self-organize and geometrically protect their macroscopic reversibility. This self-organization is not merely dynamical, it has a clear structural signature akin to the one found in a mixture of molecular liquids. Secondly, we evidence that above a maximal shaking amplitude both structural order and reversibility are lost simultaneously in the form of a 1st order non-equilibrium phase transition. Thirdly, we account for this discontinuous transition, in term of a memory-loss process.

4:53PM L7.00007 Generation of monodisperse particle-stabilized droplets with controlled particle loading at the interface , ANTHONY KOTULA, Department of Chemical Engineering, Carnegie Mellon University, CHRISTOPHER NELSON, Department of Mechanical Engineering, Carnegie Mellon University, SHELLEY ANNA, Department of Chemical Engineering & Department of Mechanical Engineering, Carnegie Mellon University — Common high-shear methods of generating particle-stablized emulsions have no direct control over the rate of droplet generation, the final droplet size distribution, or the composition of the interface, all of which are important to the interfacial and bulk rheology. In this talk, we present a method that allows for independent control over the droplet size generated and the degree of particle loading on the interface. Droplets are formed on demand at a T-junction with a desired size via a pressure-controlled generation scheme, then travel along the axis of a circular capillary containing a surface-active particle suspension. We will model the surface coverage as a function of residence time, then use light scattering methods to assess the degree of depletion of particles from the bulk and thus verify our model for two different systems: air-in-water foams stabilized by silica nanoparticles, and water droplets stabilized by silica in a continuous phase of cyclohexane. Thus, we show that by controlling channel geometry, applied inlet pressures, and residence time, we can directly control droplet size, volume fraction, and particle loading on the bubble interface, all of which are critical parameters relevant to the stability and rheology of a particle-stabilized emulsion or foam.

5:06PM L7.00008 Self assembly of droplets under shear , BINGQING SHEN, MATHILDE REYSSAT, PATRICK TABELING, ESPCI CNRS — We produce droplets of colloidal size in microfluidic systems, using step emulsification generators. The mechanism of generation allows to produce droplet clusters under control. These clusters evolve in the presence of a shear. At small shears, and for adhesive droplets, the clusters adopt equilibrium configurations that maximize the number of contact points, consistently with observations made in fluids at rest. At larger shear, we observe a rich variety of configurations, stationary, long-live or oscillatory.

5:19PM L7.00009 Drag Force on Droplet in Filtration Process , MAXIM MIKHAYLENKO, ALEX POVITSKY, The University of Akron — The drag force is studied for particles and droplets with their axis tilted with respect to Stokes flow. Boundary singularity method is adopted to solve the problem. While in stationary flow the droplet is turned so as its axis is parallel to the flow. In fiber filtration problems droplet is either attached to a fiber at an arbitrary angle between flow and fiber or is recently detached from a fiber and still turned with respect to flow. In addition, fluid droplets may form by merging of unequal droplets or form asymmetric shapes under action of gravity. In many studies Stokes force acting on arbitrary shaped particle is determined by calculating radius of spherical particle with either volume or surface same as of an original particle, and using Stokes formula. Since the approach is heuristic, the correcting coefficient is used to adhere to a particular class of particle shapes. The magnitude of drag is changed substantially with the angle. Therefore the approach of equivalent volume sphere is not valid for determination of drag for considered cases. The pressure and shear stress at the particle surface are obtained to explain the behavior of the drag with its maximum at 45 degrees. The droplets attached to a fiber are considered and the effects of fiber on the drag are studied.

5:32PM L7.00010 Hindered Brownian motion of colloidal particles near a liquid-liquid interface , WEI WANG, PETER HUANG, Binghamton University — In this work, anisotropic hindered Brownian motion of colloidal particles in the vicinity of a liquid-liquid interface is experimentally quantified and compared with established theories. Evanescent wave-based particle tracking velocimetry is used to measure the three-dimensional Brownian motion of fluorescent microspheres near an interface between water and non-polar oil. The experimental results confirm that the mobility of particles suspended in the less viscous liquid is suppressed anisotropically, and differs from the hindered mobility of particles near a solid wall. The measured hindered diffusivities are in close agreement with the theoretical predictions.

Monday, November 25, 2013 3:35PM - 4:40PM –

Session L8 Magnetohydrodynamics II 330 - Geo A. Richards, National Energy Technology Laboratory

3:35PM L8.00001 Detecting obstacles in a liquid metal flow with a small permanent magnet[1] , CHRISTIANE HEINICKE, Ilmenau University of Technology, AUNI KUNDU, University of Arizona — Flow measurement remains challenging for liquids that are opaque and chemically aggressive. Several contact-free measurement techniques have been developed that rely on the electrical conductivity of the liquid and the resulting interaction of the liquid with external magnetic fields. One of these electromagnetic techniques is the so-called Lorentz Force Velocimetry. Its advantage over other techniques is the possibility to perform measurements with a spatial resolution of the flow field. So far, however, it has remained unclear how deep the influcence of the magnetic field reaches into the liquid. We will present a liquid metal experiment whose flow structure is altered by obstacles at variable distance from the measurement device. From measurements using Lorentz Force Velocimetry we can deduce the position of objects in the liquid metal that are invisible from outside.

[1]Funded by the Helmholtz-Alliance LIMTECH, the DFG Research Training Group "Lorentz Force Velocimetry and Lorentz Force Eddy Current Testing", and the DAAD RISE program.

3:48PM L8.00002 Effects of magnetic fields on heat transfer in flowing liquid metals[1] , J. RHOADS, Princeton University, E. EDLUND, P. SLOBODA, PPPL, H. JI, Princeton University — The presence of a magnetic field can significantly change the dynamics of large and small scale features within conducting fluids. In particular, turbulent eddies with vorticity misaligned with the magnetic field are strongly damped via ohmic dissipation. Studying the anisotropic damping of the turbulence is critically important for understanding heat transport in flowing liquid metals. 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 flows with fluid velocities up to 20 cm/s and a uniform applied magnetic field strength up to 2 kG, corresponding to Reynolds numbers up to $Re \sim 10^4$ and interaction parameters up to $N \sim 10$. Heat was injected into the flow via a resistive heater placed on the free surface and the fluid temperature downstream was monitored by an array of thermocouples and an infrared camera, while an array of velocity probes provided measurements of vortical structures within the flow. The changes observed in both vortical structures and global heat transfer within the fluid will be presented.

[1]Work supported under contract DE-AC02-09CH11466.

4:01PM L8.00003 An Axisymmetric Hydromagnetic Instability in Spherical-Couette Flow , MATTHEW ADAMS, DANIEL ZIMMERMAN, University of Maryland, College Park, SANTIAGO TRIANA, Instituut voor Sterrenkunde, KU Leuven, DANIEL LATHROP, University of Maryland, College Park — We present experimental studies of the magnetized turbulent shear flow of a conducting fluid in a spherical-Couette device. Our experimental apparatus consists of an outer spherical shell concentric with an inner sphere. The geometry of the experiment makes these studies geophysically and astrophysically relevant. Liquid sodium fills the gap between the inner sphere and the shell, and we apply an axial magnetic field of varying strength. Instrumentation includes an array of hall probes to measure the induced magnetic field, providing 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 the case of corotating spheres with a rotation rate ratio of inner to outer frequency of 8, at high field an axisymmetric instability appears. This instability is anti-correlated with large fluctuations in the torque required to drive the inner sphere, indicating a significant effect on angular momentum transport. We investigate the dependence of the onset of the instability on the rotation rate ratio (or Rossby number), magnetic Reynolds number (characterizing the overall speed of the system), and Lundquist number (characterizing the strength of the applied magnetic field).

4:14PM L8.00004 Initial observations from the three meter diameter geodynamo experiment[1] , DANIEL LATHROP, DANIEL ZIMMERMAN, University of Maryland, SANTIAGO TRIANA, Institute of Astronomy at KU Leuven, HENRI-CLAUDE NATAF, L'Université Joseph Fourier, Grenoble — A liquid sodium model of the earth's outer core has been fabricated to be able to reach a magnetic Reynold number of Rm=900. The first two years of experiments were done using water as a working fluid and observed precessionally driven flows and turbulent bi-stability in spherical Couette flow. We now have the initial sodium metal flows in hand with a few months of trial runs. We have seen significant magnetic field induction by the Omega effect and many other induced magnetic field effects. While no dynamo effect has been observed at half speed (Rm $<$ 450) we have seen a gain of seven in the Omega effect, but not yet enough conversion of toroidal to poloidal field to self-generate. We have also characterized the power input of the system as a function of Rossby number, observed a dozen different non-dynamo states, and examined the fluctuations in induced magnetic field. For now, this is at parameters not yet accessible by simulation, but the observations are likely amenable to theory in reduced models.

[1]We gratefully acknowledge funding from the National Science Foundation Earth Sciences Instrumentation and Geophysics programs.

4:27PM L8.00005 Non-Darcy Effects in Magnetohydrodynamic Natural Convection in a Cavity Filled with a Porous Medium , AMAR K C, ABHILASH J. CHANDY, Department of Mechanical Engineering, University of Akron, Akron, OH - 44325-3903 — Magnetohydrodynamic (MHD) natural convection in a porous medium has received considerable attention recently, on account of its applicability in many fields of science and engineering like geophysics, chemical processes and material processing. Low-magnetic Reynolds number (Re_m) MHD natural convection is investigated in a rectangular cavity with isothermal walls on the left and right and adiabatic walls on the top and bottom. The validity of Darcy's law is addressed for high-Rayleigh number (Ra) flows with high permeability, where the velocity-pressure gradient relationship transitions from linear (that is the Darcy law) to nonlinear, due to the fact that the form drag due to solid obstacles is now comparable with the surface drag due to friction, which in turn results in the Darcy-Forchheimer law. In addition, the effect of different magnetic field strengths in terms of Hartmann numbers (Ha) is also investigated to analyze how the flow and thermal characteristics in a porous medium are influenced by the applied magnetic field.

Monday, November 25, 2013 3:35PM - 5:45PM –

Session L9 Instability: Interfacial and Thin-Film V 333 - Dan Lathrop, University of Maryland

3:35PM L9.00001 Acoustic Coupling to Kelvin-Helmholtz Instability at a Discontinuity Layer of Zero and Finite Thickness and Viscosity , ORLANDO UGARTE, V'YACHESLAV AKKERMAN, West Virginia University — The analytical formulation of Funada and Joseph [J. Fluid.Mech. 445 (2001) 263] on the Kelvin-Helmholtz (KH) instability developing at a surface separating two fluids is extended to the event of imposed acoustics field by means of the Bychkov method [Phys. Fluids 11 (1999) 3168]. Specifically, acoustical modification, mitigation and stabilization of the KH instability as well as the excitation of the parametric instability by sound waves are considered. The limits for stable/unstable regimes as a function of hydrodynamic and acoustic parameters are determined considering a linear dispersion relation for the perturbed interface. Two interacting modes are of particular interest: resonant and parametric, characterized by their frequency in relation to the disturbance oscillation. We start with an infinitely thin approach of the discontinuity surface, which is subsequently extended to a finite thickness layer (i.e. continuous velocity and density gradients are considered). A parametric study of the influence of surface tension and viscosity to the KH-acoustic coupling and stability limits is also performed.

3:48PM L9.00002 Modeling air-driven flow of a viscous film coating the interior of a rigid, vertical tube[1] , REED OGROSKY, University of Wisconsin, ROBERTO CAMASSA, GREG FOREST, JEFFREY OLANDER, University of North Carolina — The upwards, air-driven flow of a viscous fluid film coating the interior of a rigid, vertical tube is studied theoretically and numerically. The free surface of the film develops instabilities due to the interplay between interfacial stress from the airflow and surface tension from azimuthal curvature. Simple closure models for turbulent airflow coupled to long-wave asymptotic models for the liquid film have been shown to reproduce qualitatively the dynamics of the instabilities past initial transients observed in experiments. However, quantitative agreement requires improving the turbulent airflow modeling beyond leading order theories of free surface stress. An attempt in this direction is described here; the resulting model is compared with others in the literature and with experiments, for the case where the free surface is replaced by a rigid, wavy wall. This comparison is made for both wavy pipe and wavy channel flows, and the mean stress is seen to be out of phase with the wavy wall itself by a phase shift dependent on both the Reynolds number and the amplitude of the wall modulations. The free surface model is then studied through linear stability analysis and numerical solutions, both of which show improved agreement with experiments.

[1]We gratefully acknowledge support from NSF RTG DMS-0943851 and NIEHS 534197-3411

4:01PM L9.00003 Studying gas-sheared liquid film in horizontal rectangular duct with laser-induced fluorescence technique[1] , ANDREY CHERDANTSEV, Kutateladze Institute of Thermophysics, Novosibirsk, Russia, DAVID HANN, BARRY AZZOPARDI, University of Nottingham — High-speed LIF-technique is applied to study gas-sheared liquid film in horizontal rectangular duct with 161 mm width. Instantaneous distributions of film thickness resolved in both longitudinal and transverse coordinates were obtained with a frequency of 10 kHz and spatial resolution from 0.125 mm to 0.04 mm. Processes of generation of fast and slow ripples by disturbance waves are the same as described in literature for downwards annular pipe flow. Disturbance waves are often localized by transverse coordinate and may have curved or slanted fronts. Fast ripples, covering disturbance waves, are typically horseshoe-shaped and placed in staggered order. Their characteristic transverse size is of order 1 cm and it decreases with gas velocity. Entrainment of liquid from film surface can also be visualized. Mechanisms of ripple disruption, known as "bag break-up" and "ligament break-up," were observed. Both mechanisms may occur on the same disturbance waves. Various scenarios of droplet deposition on the liquid film are observed, including the impact, slow sinking and bouncing, characterized by different outcome of secondary droplets or entrapped bubbles. Number and size of bubbles increase greatly inside the disturbance waves. Both quantities increase with gas and liquid flow rates.

[1]EPSRC Programme Grant MEMPHIS (EP/K003976/1), and Roll-Royce UTC (Nottingham, for access to flow facility)

4:14PM L9.00004 Wavy liquid films in interaction with a strongly confined laminar gas flow: Modeling and direct numerical simulations[1] , GEORG F. DIETZE, CHRISTIAN RUYER-QUIL, Laboratoire FAST, UMR 7608, Batiment 502, Campus universitaire, 91405 Orsay — Different technological settings concern the flow of a wavy liquid film in contact with a strongly confined gas flow. Micro-gaps for instance, which are employed for the cooling of electronic equipment, involve a pressure-driven evaporating liquid film flowing co-currently to its own vapor. In packed columns used for distillation, falling liquid films sheared by a counter-current gas flow occur within narrow channels. Surface waves on the liquid-gas interface of these flows play an important role as they intensify scalar transfer and may cause flooding of the channel. However, their accurate prediction by full numerical simulation is associated with a substantial computational cost. We evaluate an alternative approach based on a low-dimensional integral boundary layer formulation applied to both fluid layers. The resulting model captures the long-wave (Yih and Kapitza) instabilities of the flow accurately and allows calculations on long domains at low computational cost. These evince a number of intricate wave-induced flow structures within the film and gas as well as a possible route to the flooding of narrow channels under counter-current gas flow conditions. Comparisons with direct numerical simulations using the VOF-CSF approach as well as experiments are convincing.

[1]GFD acknowledges support from DAAD (Deutscher Akademischer Austauschdienst)

4:27PM L9.00005 Liquid falling films: linear stability and direct numerical simulation[1] , PATRICK SCHMIDT, University of Edinburgh, LENNON O'NARAIGH, University College Dublin, PRASHANT VALLURI, MATHIEU LUCQUIAUD, University of Edinburgh — Interfacial instability of falling liquid films in counter-current contact with a turbulent gas phase is investigated by means of an Orr-Sommerfeld analysis. This study is complemented by a full energy budget analysis, identifying the key mechanisms of the instability. This gives first insight into the dynamic behaviour of the two-phase system, which is relevant for a wide range of technical applications, such as absorption and distillation. The linear stability analysis is also used to identify the operating limits of a counter-current operation i.e. the so-called loading and flooding limits. In addition, the results of this analysis are benchmark for direct numerical simulations using the newly launched Two-Phase Level Set (http://sourceforge.net/projects/tpls/) solver. High resolution DNS is used to obtain detailed knowledge of important mechanisms at play, especially with regard to interfacial instability and transient system behaviour, which can help to design more efficient mass transfer equipment such as structured packings.

[1]Sulzer Chemtech Ltd, EPSRC, Energy Technology Partnership

4:40PM L9.00006 An experimental investigation of fingering instability and growth dynamics in inclined counter-current gas-liquid channel flow , JORDAN PURVIS, RAVI MISTRI, CHRISTOS MARKIDES, OMAR MATAR, Imperial College London — The results of an experimental study involving low Reynolds number, counter-current flows of glycerol and air on an inclined glass substrate inside a rectangular channel are presented. The interface forms a thickened front immediately upstream of a thin, precursor layer region. This front is vulnerable to spanwise perturbations which, under certain conditions, grow to acquire the shape of "fingers." Decreasing the inclination angle has a stabilizing effect on the front: complete stability is achieved below a critical angle whose value depends on the remaining system parameters. Regions of transient finger formation are also observed. It is also found that increasing the ratio of the precursor to the inlet film thickness, and increasing the liquid and air flow-rates also exerts a stabilizing effect on the interface. Analyses of the initial finger growth-rate corroborate the findings of previous theoretical work, showing this growth-rate to be independent of inclination angle and liquid film Reynolds number, and weakly-dependent on the air flow-rate for low inclination angles.

4:53PM L9.00007 Absolute and convective instabilities in turbulent gas-laminar liquid film flows , RAJAGOPAL VELLINGIRI, Department of Chemical Engineering, Imperial College London, UK, DMITRI TSELUIKO, Department of Mathematical Sciences, Loughborough University, UK, SERAFIM KALLIADASIS, Department of Chemical Engineering, Imperial College London, UK — Gas-liquid flows are important from a fundamental fluid mechanics point of view, but are also central in a variety of engineering applications, such as distillation, absorption and cooling of electronic devices. Our prototypical system for such flows consists of a thin laminar liquid film flowing down an inclined plate in the presence of a countercurrent turbulent gas. The liquid flow is influenced by the gas through the tangential and normal stresses acting at the interface. We develop low-dimensional models for the liquid-flow problem: a long-wave model and a weighted integral-boundary layer (WIBL) model. These models, along with the Orr-Sommerfeld problem derived from the full Navier-Stokes equations and associated boundary conditions are used to explore the linear stability of the liquid-gas system. For a given liquid flow rate, we show that the wave velocity decreases with increasing gas shear before changing direction at the "flooding point." The appearance of this point is linked to the onset of absolute instability, where a localized disturbance gets amplified and contaminates the whole domain. This is also marked by the collision of two spatial branches at a saddle point. We supplement our stability analysis with time-dependent computations of the WIBL model.

5:06PM L9.00008 Linear stability analysis of thin films in wall bounded shear flow , AHMED KAFFEL, AMIR RIAZ, University of Maryland — In this study we examine the stability of core annular flow of two fluids with large density and viscosity ratios to investigate the physical mechanisms associated to thin liquid films flow in microgap channels. Emphasis will be placed on predicting and controlling the growth of interfacial instabilities which can lead to the rupture of the thin liquid films encountered in annular flows. A multi-domain Chebyshev collocation spectral method along with QZ eigenvalue solver are used to solve the Orr-Sommerfeld stability equations in both layers. The algorithm is computationally efficient and accurate in reproducing the whole spectrum of the eigenvalues and associated eigenfunctions. The derivation of the asymptotics of these modes shows that the numerical eigenvalues are in agreement with the analytic formula obtained previously by Yih (1967), Orszag (1971), Higgins et al (1988), Dongarra (1996) and Sahu et al (2007). The numerical simulations and experiments are carried out to quantify unstable wave patterns with respect to the underlying fluid dynamic mechanism for various flows rates. We consider the case of isothermal, non-adiabatic, parallel flow of liquid and vapor phases. A parametric study is analyzed and the numerical stability results are presented and will be used later as a tool to validate the direct numerical solver and to identify the physical mechanisms in two-phase liquid vapor flows.

5:19PM L9.00009 Electrostatic control of flows of moderate Reynolds number[1] , DEMETRIOS PAPAGEORGIOU, ALEX WRAY, OMAR MATAR, Imperial College London — Film flow down an inclined plane is a widely investigated problem because it serves as an important prototypical situation for analysis. Under the assumption that the characteristic wavelength of coherent structures is long relative to the thickness of the film, this system can be modelled to second order by the boundary-layer equations. However, the perturbative approach, which enslaves the system to the dynamics of the interface, typically results in equations (e.g. the Benney equation) that exhibit finite-time blow-up. It has been shown that a weighted-residual approach gives rise to simple equations which exhibit very good agreement with both direct numerical simulations and experiments in both the drag-gravity and drag-inertia regimes. We extend this system by allowing the plane to serve as an electrode, and incorporating a second parallel plane positioned above the fluid. The variation in the resultant electric fields in each region induces a Maxwell stress at the interface. We validate our model in one dimension via comparisons of linear theory, and by direct numerical simulations of both transient solutions and traveling waves. We then extend this to the two dimensional case to exhibit the degree of control afforded by the electric field.

[1]EPSRC DTG

5:32PM L9.00010 FRAP in thin film flows , JASON WEXLER, IAN JACOBI, HOWARD STONE, Princeton University — A new technique is proposed for measuring the velocity field within thin liquid films, which combines Fluorescence Recovery After Photobleaching (FRAP) measurements with two-dimensional Taylor dispersion analysis. FRAP is a technique used largely by biologists to measure the diffusion coefficient of compounds in living cells. A small spot of fluorescent dye is bleached and then monitored for subsequent fluorescence recovery. The rate of recovery can be related to the coefficient of molecular diffusion. In our experiments we apply FRAP to a flowing liquid film, where advection, in addition to molecular diffusion, contributes to the evolution of the bleached spot. By employing simple optical measurements of the rate of advection and diffusion, combined with an analysis of dispersion, we can uniquely determine the velocity profile within a thin film. As a proof of concept we apply this technique to shear-driven flow over a liquid film within a micro-patterned surface.

Monday, November 25, 2013 3:35PM - 5:45PM –

Session L10 Instability: General II 334 - Outi Tammisola, University of Cambridge

3:35PM L10.00001 Oscillations in Power and Structure During the Transition to Defect Turbulence , MARCUS DAUM, ZRINKA GREGURIC FERENCEK, JOHN CRESSMAN, George Mason University — Electroconvecting liquid crystals support a wide range of dynamics from ordered rolls to highly chaotic states characterized by the creation, interactions, and dissipation of defects. In addition to visual observations these systems also allow the direct measurement of the electrical power injected into them. Here we report on a remarkable phenomenon that occurs when a sample is abruptly driven from an ordered steady state to a complex driven state. During such transitions the system transiently maintains its ordered structure beyond the transition to defect dynamics. The order enables the system to absorb more and then less power than in the steady state. By simultaneously imaging the system and measuring the power injected into the system, we are able to investigate the relationship between defect dynamics, conductivity, and power injection in this system.

3:48PM L10.00002 Numerical Simulation of Liquid Sheet Instability in a Multiphase Flow Domain , CHATTERJEE SOUVICK, Virginia Tech, SOUMIK MAHAPATRA, Jadavpur University, ACHINTYA MUKHOPADHYAY, IIT Madras, SWARNENDU SEN, Jadavpur University — Instability of a liquid sheet leading to the formation of droplets is a classical problem finding a wide range of multi-scale applications like gas turbine engines and inkjet printers. Numerical simulation of such a phenomenon is crucial because of its cost and time effective nature. In this work, the hydrodynamics in a custom designed nozzle is analyzed using Volume of Fluid method in Ansys Fluent. This innovative nozzle design includes an annular liquid sheet sandwiched between two air streams such that the inner air channel is recessed to a certain length. Such a recession leads to interaction between the two multiphase streams inside the atomizer resulting to an increased shear layer instability which augments the disintegration process. The numerical technique employed in this work couples Navier Stokes equation with VoF surface tracking technique. A parametric study with the hydrodynamic parameters involved in the problem, as well as the recession length, is performed while monitoring the axial and tangential exit velocities along with the spray cone angle. Comparison between the full 3D model and two different equivalent 2D axisymmetric models have been shown. The two axisymmetric models vary based on conserving different physical parameters between the 2D and 3D cases.

4:01PM L10.00003 A novel numerical approach and stability analysis of thermo-acoustic phenomenon in the Rijke tube problem , TARANEH SAYADI, Laboratoire d'Hydrodynamique, Ecole Polytechnique, VINCENT LE CHENADEC, EM2C Laboratory, Ecole Centrale Paris, PETER SCHMID, Laboratoire d'Hydrodynamique, Ecole Polytechnique, FRANCK RICHECOEUR, MARC MASSOT, EM2C Laboratory, Ecole Centrale Paris — The modeling of thermo-acoustic coupling in reactive flows represents a challenging task. In this study, we focus on the Rijke tube problem, which includes relevant features such as a compact acoustic source, an empirical modeling of the heat source, and non-linearities. This thermo-acoustic system features a complex dynamical behavior, which renders the characterization of the different encountered flow regimes difficult. In order to synthesize accurate time series, we tackle this problem from a numerical point-of-view, and start by proposing a dedicated solver designed for dealing with the underlying stiffness, in particular, the retarded time and the discontinuity at the location of the heat source. Convergence and parametric studies are carried out to assess the accuracy of the discretization, hence laying a foundation for a stability analysis of the semi-discrete system. This stability analysis is performed by means of the projection method proposed by Jarlebring [1], which alleviates the linearization of the retarded term, and is used to validate the numerical results. Finally, the focus is set on the application of the dynamic mode decomposition [2] technique to study bifurcations.

[1] Jarlebring, E., Thesis, 2008
[2] Schmid P., JFM, 2010

4:14PM L10.00004 Nonlinear optimization of multiple perturbations and stochastic forcing of subcritical ODE systems , DANIEL LECOANET, UC-Berkeley, RICH KERSWELL, Bristol University — Subcritical transition between states has been proposed to explain a variety of phenomena, including transition to turbulence in shear flows, and the generation of a magnetic dynamo in accretion discs. These systems feature an easily specified linearly stable "laminar" equilibrium state, along with at least one other stable "turbulent" state. We present simple 2D ODE model systems with these features, and study how the systems react to different types perturbations. First, we extend variational techniques used to study transition to turbulence in shear flows (e.g. Pringle & Kerswell 2010, Rabin et al 2012) to find the optimal (that is, closest to the "turbulent" state at late time) set of multiple perturbations, each occurring at a different time, that will cause a transition to the "turbulent" state. We find that some systems transition to the "turbulent" state much more easily with multiple perturbations than with a single perturbation. Second, we introduced random noise into the model systems, and determined the mean exit time from the attractor of the "laminar" state. We find that systems in which "turbulence" is more easily triggered by multiple perturbations have shorter mean exit times when subjected to noise.

4:27PM L10.00005 Second-order sensitivity of eigenvalues: large or spanwise wavy perturbations , OUTI TAMMISOLA, University of Cambridge, FLAVIO GIANNETTI, VINCENZO CITRO, University of Salerno, MATTHEW JUNIPER, University of Cambridge — Sensitivity maps can be used to determine how the stability of a flow changes with control (by for example a control cylinder), or changes in the flow parameters. However, being linear with respect to the control parameter, the sensitivities can only represent the influence of small-amplitude control. More importantly, the sensitivities vanish for some important classes of perturbations, such as spanwise wavy base flow modifications. Spanwise wavy modifications can appear in a flow due to inflow asymmetry or streakiness. In flow control, spanwise wavy steady blowing and suction has been shown to suppress vortex shedding behind a cylinder in computations at Re=140 (Kim & Choi, PoF 2005, 17, 033103). Sensitivities can be derived from a standard perturbation analysis. In this study, we generalize the sensitivities by considering the second-order term in the perturbation expansion. We derive some general insights about the effects of large and wavy base-flow modifications from an expansion in the eigenmode basis. As an example of the "second-order sensitivity," we consider the effect of steady streakiness on the global instability of a backwards-facing step.

4:40PM L10.00006 Unsteady engulfment regime in a three dimensional T-mixer: stability and sensitivity analyses , SIMONE CAMARRI, DICI, University of Pisa, ANDREA FANI, University of Nice, MARIA VITTORIA SALVETTI, DICI, University of Pisa — Micro T-mixers are important devices in microfluidics; for instance, they are often used as junction elements in complex micro-systems. Most of the studies in the literature focused their attention on the steady engulfment regime, characterized by a loss of the flow symmetries in the outflow channel which in turn leads to a considerable increase of the mixing efficiency. It has been recently observed that if the Reynolds number is increased beyond the steady engulfment critical value, the flow may become unsteady with a periodic pulsating behavior and this regime corresponds to a significant further increase of mixing compared to the steady one. We consider a given T-mixer geometry and we combine direct numerical simulations with fully 3D linear stability and sensitivity analyses to characterize the unsteady engulfment regime in terms of critical Reynolds number, characteristic time frequencies and flow dynamics. The unsteadiness seems to be triggered by a critical value for the intensity and orientation of vortices at the confluence in the mixing channel; the instability core is indeed located in the center of these vortices. The sensitivity to a generic modification of the base-flow is investigated, to obtain indications on possible control strategies.

4:53PM L10.00007 Short-wave analysis of 3D and 2D instabilities in a driven cavity , PAOLO LUCHINI, FLAVIO GIANNETTI, VINCENZO CITRO, Università di Salerno - DIIN — The short-wave asymptotic approximation of inviscid instabilities proposed by Bayly (*Phys. Fluids* **31**, 1988) and Lifschitz & Hameiri (*Phys. Fluids A* **3**, 1991) is here applied to the dominant (three-dimensional) instability of two-dimensional flow in either an open or a closed driven cavity, and compared to the structural sensitivity obtained by direct-adjoint computation. The comparison shows that the structural sensitivity of the eigenmode is indeed localized around the critical streamline identified by short-wave asymptotics, and that the latter provides a reasonably good expression of even the first unstable eigenvalue at critical Reynolds number. Curiously enough, the same approximation appears also to apply with success to the two-dimensional instability of the same flow, despite the absence of a large spanwise wavenumber to be used as an expansion parameter. The theoretical justification of this extension, and the importance of phase quantization along the trajectory, will be discussed.

5:06PM L10.00008 Effect of an axial flow on three-dimensional instabilities in Stuart vortices[1] , MANIKANDAN MATHUR, Department of Aerospace Engineering, IIT Madras, Chennai, India, SABINE ORTIZ, LadHyX, Ecole Polytechnique, Palaiseau, France, THOMAS DUBOS, Laboratoire Meteorologie Dynamique, Ecole Polytechnique, Palaiseau, France, JEAN-MARC CHOMAZ, LadHyX, Ecole Polytechnique, Palaiseau, France — In this talk, we present a stability analysis of the Stuart vortices in the presence of an axial flow by numerically solving the local stability equations derived by Lifschitz & Haimeri (1991). Deriving the criteria for wave vectors to be periodic upon their evolution around flow trajectories that are periodic in a plane perpendicular to the axial direction, we integrate the amplitude equations around periodic trajectories for periodic wave vectors. The elliptic and hyperbolic instabilites, which are present without the axial velocity, disappear beyond a threshold value for the axial velocity strength. Furthermore, a threshold axial velocity strength, above which a new centrifugal instability branch is present, is identified. A heuristic novel criterion, which reduces to the Leibovich & Stewartson (1983) criterion in the limit of an axisymmetric vortex, for centrifugal instability in a non-axisymmetric vortex with an axial flow is then proposed and validated.

[1]We acknowledge funding from Agence Nationale de la Recherche grant ANR 09-JCJC- 0108-01 for this work.

5:19PM L10.00009 The rich life of light rising spheres , JACQUES MAGNAUDET, IMFT/CNRS, ALICE LIEU, FRANCK AUGUSTE, IMFT — The straight path of spheres falling or rising in a weakly viscous fluid is known to become unstable beyond a critical value of the so-called Archimedes number Ar, a Reynolds number built on the gravitational velocity scale. Various styles of non-vertical paths have been reported so far: steady or oscillating oblique, planar zigzags, three-dimensional chaotic, etc. However despite careful computations and experiments, there is currently no consensus as regards the possible critical density ratio m* below which significant departures from straight (vertical or oblique) path are observed. To revisit this question, we carry out a detailed DNS study focused on rising spheres (m*<1) in the range $150 \leq Ar \leq 350$. Non-straight paths with significant horizontal excursions are observed throughout the whole range of m*. In addition to the various aforementioned types of paths we also identify other styles such as intermittent zigzagging/oblique paths and find that very light spheres describe highly nonlinear zigzags and have drag coefficients up to 15% beyond standard values.

5:32PM L10.00010 Stability of Resting Cylinders , CUNJING LU, CHRISTOPHE CLANET, DAVID QUERE, PMMH, CNRS, ESPCI, Paris France - Ladhyx, CNRS, Ecole Polytechnique, Palaiseau, France — The capillary instability of a cylinder is a classical topic in the field of fluid interface. As experimentally found by Plateau, the instability happens if the ratio of the wavelength of an axisymmetric fluctuation to the initial diameter of the cylinder is larger than 3.14. We discuss how the fact that the cylinder rests on a superhydrophobic surface (which avoids stabilizing) pining effects modifies this picture. By employing a finite element method, we mainly conclude that: (1) the ratio of the wavelength to the width of the liquid cylinder increases as the liquid cylinders grow; (2) above a critical value D_c of the cylinder diameter, the instability disappears; (3) conversely, decreasing the cylinder diameter restores the instability, yet at a wavelength larger than the Plateau value. This is attributed to the loss of axisymmetry, and discussed more generally by considering the effect of confinement around the cylinder.

Monday, November 25, 2013 3:35PM - 5:45PM –

Session L11 Non-Newtonian Flows I 335 - Amy Shen, University of Washington

3:35PM L11.00001 Worst-case amplification of disturbances in inertialess shear-driven flows of viscoelastic fluids , MIHAILO JOVANOVIC, BINH LIEU, SATISH KUMAR, University of Minnesota — Amplification of deterministic disturbances in inertialess shear-driven channel flows of viscoelastic fluids is examined by analyzing the frequency responses from spatio-temporal body forces to the velocity and polymer stress fluctuations. In strongly elastic flows, we show that disturbances with large streamwise length scales may be significantly amplified even in the absence of inertia. For fluctuations without streamwise variations, we derive explicit analytical expressions for the dependence of the worst-case amplification (from different forcing to different velocity and polymer stress components) on the Weissenberg number (We), the maximum extensibility of the polymer chains (L), the viscosity ratio and the spanwise wavenumber. For the Oldroyd-B model, the amplification of the most energetic components of velocity and polymer stress fields scales as We^2 and We^4. On the other hand, the finite extensibility of polymer molecules limits the largest achievable amplification even in flows with infinitely large Weissenberg numbers: in the presence of wall-normal and spanwise forces the amplification of the streamwise velocity and polymer stress fluctuations is bounded by quadratic and quartic functions of L.

3:48PM L11.00002 Linear stability analysis of the stick-slip flow of a viscoelastic fluid following the Phan-Thien Tanner model[1] , JOHN TSAMOPOULOS, University of Patras, GEORGE KARAPETSAS, University of Thessaly — It is well known that during extrusion of viscoelastic fluids various flow instabilities may arise resulting in a distorted free surface. In order to investigate the factors generating these instabilities we perform a linear stability analysis at zero Reynolds number around the steady solution of the cylindrical or planar stick-slip flow for a viscoelastic fluid following the PTT model. The stick-slip flow is an important special case of the extrudate swell problem, since the latter reduces to it in the limit of infinite surface tension. We will show that the flow becomes unstable as the Weissenberg number increases above a critical value, due to a Hopf bifurcation suggesting that the flow will become periodic in time. Both the critical value of the Weissenberg number and the frequency of the instability depend strongly on the rheological parameters of the viscoelastic model. The elasticity alone can be responsible for the appearance of instabilities in the extrusion process of viscoelastic fluids and the often used assumptions of wall slip or compressibility, although they might be present, are not required. Finally, the mechanisms that produce these instabilities are examined through energy analysis of the disturbance flow.

[1]The authors would like to acknowledge the financial support by the General Secretariat of Research and Technology of Greece under the Action "Supporting Postdoctoral Researchers" (Grant No: PE8/906), and under the "Excellence Program" (Grant No: 1918)

4:01PM L11.00003 Localized disturbances in channel flow of a viscoelastic fluid , AKSHAT AGARWAL, Imperial College London, LUCA BRANDT, Linne Flow Centre, SeRC, KTH Mechanics, TAMER ZAKI, Imperial College London — Linear and non-linear growth of a localized disturbance in polymeric channel flow is investigated using Direct Numerical Simulations. The polymeric stress is represented by the FENE-P model. When the amplitude of the intial disturbance is sufficiently small, the presence of the polymer reduces the linear amplification of the disturbance energy, and this stabilizing influence depends on the Weissenberg number, maximum polymer extensibility and the ratio of the solvent viscosity to the total viscosity. When the initial disturbance amplitude is increased, the same trend is identified in the early linear phase. In the subsequent non-linear phase, the behaviour of the Newtonian and polymeric flows are substantially different. In the Newtonian case, non-linear growth is followed by an ultimate decay of the disturbance energy due to viscosity. On the other hand, in the polymeric flow, the non-linear terms due to the solvent contribution are reduced. However, a new energy growth mechanism is present and leads to bypass transition.

4:14PM L11.00004 Contravariant and covariant dumbbells in polymer-diluted viscoelastic turbulence , KIYOSI HORIUTI, SHOHEI TAKEU, Tokyo Institute of Technology — We carried out numerical study to reveal the mechanism of drag reduction (DR) in polymer-diluted flows. The polymer chains are modeled as elastic dumbbells. Our aim is to elucidate the effect of non-affinity in the motion of dumbbells on DR, in which their motions do not precisely correspond to macroscopically-imposed deformation. We conduct analysis in forced homogeneous isotropic turbulence by connecting a macroscopic description (DNS) with a mesoscopic Brownian dynamics of dumbbells (BDS). The dumbbell connector vector is convected as either contravariant or covariant vectors. Contravariant dumbbells orient in the stretching direction of the strain and elasticity is incurred on the tubular structures. Covariant dumbbells orient in the direction which maximizes the stretching by the solvent deformation and direct outward perpendicularly on the planar structures. They exert an extra tension on vortex sheet, which leads to attenuation of energy cascade, resulting in a larger DR than in contravariant dumbbells. In the mixture of contravariant and covariant dumbbells, DR is intermediate between those caused in individually released cases. The two dumbbells form a unit in which contravariant dumbbell is transversely aligned with the covariant dumbbell.

4:27PM L11.00005 Turbulence in dilute polymer solutions , ALEXANDRE DE CHAUMONT QUITRY, NICHOLAS T. OUELLETTE, Yale University — Turbulence in complex fluids encompasses many fascinating phenomena, ranging from drag reduction to elastic turbulence. We focus on inertial turbulence in a dilute polymer solution in order to understand how small changes a fluid's micro-scale properties result in large-scale flow changes. While there has been considerable progress in identifying such a mechanism in wall-bounded flows, it remains unclear in unbounded flows. We use Lagrangian Particle Tracking to measure the effect of 5 p.p.m by weight of polyacrylamide in water by imaging the central region of an experimental Von Karman flow, generated by placing counter-rotating impellers in a cylindrical chamber. While the fluid's viscosity hardly departs from that of water at such low concentrations, we observe a strong suppression of velocity fluctuations in the inertial range.

4:40PM L11.00006 ABSTRACT WITHDRAWN —

4:53PM L11.00007 Suppression of the Rayleigh-Plateau instability on a vertical fibre coated with wormlike micelle solutions , FRANÇOIS BOULOGNE[1], LUDOVIC PAUCHARD, FRÉDÉRIQUE GIORGIUTTI-DAUPHINÉ, UPMC Univ Paris 06, Univ Paris-Sud, CNRS, Lab FAST Orsay France, MARC-ANTOINE FARDIN, SANDRA LEROUGE, Laboratoire Matière et Systèmes Complexes, CNRS UMR 7057, Université Paris Diderot, Paris, France — When a liquid film is coating a fibre, it undergoes spatial thickness variations due to the Rayleigh-Plateau instability. We report on the Rayleigh-Plateau instability in films of giant micelles solutions coating a vertical fibre. We observe that the dynamics of thin films coating the fibre could be very different from the Newtonian or standard Non-Newtonian cases. By varying the concentration of the components of the solutions and depending on the film thickness, we show for the first time that the Rayleigh-Plateau instability can be stabilized using surfactant solutions. Using global rheology and optical visualisations, we show that the development of shear-induced structures is required to stabilize the micellar film along the fibre. Assuming that the viscoelastic properties of the shear-induced state can be described by a simple model, we suggest that, in addition to the presence of shear-induced structures, the latter must have an elastic modulus greater than a critical value evaluated from a linear stability analysis. Finally, our analysis provides a way of estimating the bulk elasticity of the shear-induced state.

[1]Current address: Complex fluids group, Mechanical and Aerospace Engineering, Princeton University (USA)

5:06PM L11.00008 Irreversible Gelation in Wormlike Micellar Solutions via Microfluidics , JOSHUA CARDIEL, YA ZHAO, PERRY CHEUNG, AMY SHEN, University of Washington — Surfactant molecules can self-assemble into various morphologies under proper combinations of ionic strength, temperature, and flow conditions. At equilibrium, the wormlike micelles can transition from entangled to branched and multi-connected structures with increasing salt concentration. Under specific flow conditions, micellar structure transition can follow different trajectories. In this work we consider the flow of two semi-dilute wormlike micellar solutions through microposts, focusing on their microstructural and rheological evolution. Both solutions contain cetyltrimethylammonium bromide (CTAB) and sodium salicylate (NaSal). One is weakly viscoelastic and shear thickening while the other is strongly viscoelastic and shear thinning. When subject to strain rates $\sim 10^3$ s^{-1} and strain $\sim 10^3$, we observe irreversible gelation, with entangled, branched, and multi-connected micellar bundles, evidenced by electron microscopy. We also show that the rheological properties of the shear-thickening precursor are smaller than those of the gel, while the rheological properties of the shear-thinning precursor are several times larger than those of the ge. This rheological property variation is associated with their respective structural evolution.

5:19PM L11.00009 On the origin and evolution of streaks in polymeric shear flows , JACOB PAGE, TAMER ZAKI, Imperial College London — Streaks are a ubiquitous feature in transitional shear flows of both Newtonian and complex fluids. A model problem is formulated where streaks are generated in response to forcing by a decaying streamwise vortex in an Oldroyd-B fluid, and the effects of inertia and elasticity are examined. The dynamics are found to be largely governed by a single parameter: the ratio of the solvent diffusion to the polymer relaxation timescales. When the time scales are disparate, the "quasi-Newtonian" and "elastic" dynamics can be distinguished. The "quasi-Newtonian" evolution of the streaks in the polymeric flow matches the Newtonian equivalent at the same total (solvent) Reynolds number when polymer relaxation is very fast (slow). The "elastic" response is significant, when the polymer relaxation time is long, and leads to significant streak amplification even with very weak inertia. When the diffusion and polymer relaxation timescales are commensurate, the streaks are re-energised in a periodic cycle. This behaviour is enhanced in the instantaneously elastic limit where the governing equation reduces to a wave equation with harmonic forcing. The streak re-energisation is demonstrated to be a superposition of trapped inertio-elastic shear waves.

5:32PM L11.00010 Intermittent Flow In Yield Stress Fluids Slows Down Chaotic Mixing , JALILA BOUJLEL, DAWN WENDELL, EMMANUELLE GOUILLART, FRANCK PIGEONNEAU, PIERRE JOP, Laboratoire Surface du Verre et Interfaces, LABORATOIRE SURFACE DU VERRE ET INTERFACES TEAM — Many mixing situations involve fluids with non-Newtonian properties: mixing of building materials such as concrete or mortar are based on fluids that have shear- thinning rheological properties. Lack of correct mixing can waste time and money, or lead to products with defects. When fluids are stirred and mixed together at low Reynolds number, the fluid particles should undergo chaotic trajectories to be well mixed by the so-called chaotic advection resulting from the flow. Previous work to characterize chaotic mixing in many different geometries has primarily focused on Newtonian fluids. First studies into non-Newtonian chaotic advection often utilize idealized mixing geometries such as cavity flows or journal bearing flows for numerical studies. Here, we present experimental results of chaotic mixing of yield stress fluids with non-Newtonian fluids using rod-stirring protocol with rotating vessel. We describe the various steps of the mixing and determine their dependence on the fluid rheology and speeds of rotation of the rods and the vessel. We show how the mixing of yield-stress fluids by chaotic advection is reduced compared to the mixing of Newtonian fluids and explain our results, bringing to light the relevant mechanisms: the presence of fluid that only flows intermittently, a phenomenon enhanced by the yield stress, and the importance of the peripheral region. This result is confirmed via numerical simulations.

Monday, November 25, 2013 3:35PM - 5:32PM –

Session L12 Vortex Dynamics and Vortex Flows VII 336 - Jonathan Naughton, University of Wyoming

3:35PM L12.00001 Suppression of vortex-induced vibrations in a flexible cylinder with elastic splitter plates[1] , FRANCISCO HUERA-HUARTE[2], Universitat Rovira i Virgili — Suppression of vortex-induced vibrations (VIV) is a topic that has received a lot of attention due to its practical implications in engineering design. Experiments have been conducted in a recirculating free surface water channel, with a working section of dimensions 1x1.1x2.5 m. A cylinder model made of a spring and a plastic cover was used for the experiments. It was placed horizontally and fully submerged in the water channel's free stream, hanging from two submersible load cells arranged to measure the total drag force on the cylinder. The model had several white points painted on its surface, so its VIV motion was obtained by imaging it with two cameras synchronised with a strobe light. Image processing allowed to obtain the displacements along the length of the cylinder with sub-pixel accuracy. Digital Particle Image Velocimetry (DPIV) was also used to quantify the wake downstream the cylinder. A full set of experiments was made for reference purposes with a plain cylinder without suppressors, and for the same conditions, several passive suppression devices such as elastic splitter plates of different sizes and shapes, were installed on the cylinder. Passive VIV suppression with drag reduction was achieved with some of the configurations tested.

[1]Funding provided by the Spanish Ministry of Science through grant DPI2012-37904 is acknowledged

[2]Department of Mechanical Engineering

3:48PM L12.00002 Harbor seal whiskers synchronize with frequency of upstream wake , HEATHER BEEM, MICHAEL TRIANTAFYLLOU, MIT — Harbor seals are able to use their whiskers to track minute water movements, such as those left in the wake of a fish [1]. The current study is a simple representation of what the whiskers experience as the seal chases a fish. A scaled whisker model (average cross-flow diameter: d_w) is first tested in a towing tank by itself and then towed behind a larger cylinder ($d_c = 2.5d_w$), which serves as a wake generator. A flexing plate attached to the model base allows the whisker to freely vibrate in response to the flow. Measurements from strain gages on the plate are calibrated to tip deflections. While in the cylinder wake, the whisker vibrates with an amplitude up to ten times higher than it does on its own ($A/d_w = 0.15$). Also, the whisker synchronizes with the vortex shedding frequency ($f_s = \frac{0.2U}{d_c}$) of the upstream cylinder over the range of reduced velocities tested, whereas on its own, the whisker oscillates around its own natural frequency in water. Seals may use the difference in vibration amplitude and frequency between these two cases to help detect the presence of a vortex wake.

[1] Dehnhardt, G., et al. (1998). "Seal whiskers detect water movements," Nature, 394(6690), 235-6.

4:01PM L12.00003 Vortex-Induced Vibration (VIV) Reduction Properties of Seal Whisker-Like Geometries[1] , HENDRIK HANS, Nanyang Technological University, Singapore-MIT Alliance for Research and Technology, JIANMIN MIAO, Nanyang Technological University, MICHAEL TRIANTAFYLLOU, Massachusetts Institute of Technology — Biological studies have shown that harbor seal whiskers are capable of reducing Vortex-Induced Vibrations (VIV). As the whiskers have convoluted geometry, it is necessary to evaluate the parameters that define their VIV reduction properties. Whisker-Like Geometries (WLGs) consisting of all but one feature on the true whisker geometry are designed. Comparison of VIV on these WLGs with VIV on circular and elliptical cylinders at Re = 500 is performed. Three-dimensional simulations of flow past these geometries, which are allowed to freely vibrate in crossflow, are performed with the Implicit Large Eddy Simulation as the turbulence model. The results indicate that the existence of axial undulations is the most dominant feature that affects the VIV reduction. The smallest VIV is observed on WLGs with dual-axial undulations and the largest VIV is observed on the circular cylinder. Variations in the features of the WLGs result in noticeable changes in their VIV. The circular cylinder is observed to response as a steady system while the WLGs with dual-axial undulations are observed to respond as a chaotic system. The response of WLGs with single-axial undulations is found to depend on their detailed features.

[1]I would like to acknowledge the support and funding from National Research Foundation (NRF) through CENSAM of Singapore-MIT Alliance for Research and Technology and Nanyang Technological University

4:14PM L12.00004 Higher Harmonic Forces in Purely Crossflow Vortex-Induced Vibrations , YAHYA MODARRES-SADEGHI, BANAFSHEH SEYED-AGHAZADEH, University of Massachusetts, Amherst, REMI BOURGUET, Institut de Mécanique des Fluides de Toulouse, GEORGE KARNIADAKIS, Brown University, MICHAEL TRIANTAFYLLOU, MIT — In vortex-induced vibrations (VIV) of flexibly-mounted rigid cylinders free to oscillate both in the inline and crossflow directions, higher (3rd) harmonic forces have already been observed in the crossflow direction. In the present work, we report higher harmonic force components for a flexibly-mounted rigid cylinder with only one degree of freedom in the crossflow direction. We show that the inline displacement is not necessary to observe higher harmonic components in the crossflow force spectrum. Due to the relative velocity of the cylinder with respect to the oncoming flow, the lift and drag forces make an angle with respect to the crossflow and inline directions, and the contribution of the components of each of these forces in the crossflow direction results in a 3rd harmonic force component. These higher harmonic components have been observed in self-excited VIV experiments, performed in a water tunnel for a Reynolds number range of Re=400-1000, as well as in numerical simulation results at Re=100. We also find that the maximum ratio of the 3rd harmonic to the 1st harmonic occurs when the phase between the crossflow force and displacement changes from 0 to 180 degrees, resulting in a small first harmonic component.

4:27PM L12.00005 Flow-induced vibrations of a rotating cylinder , REMI BOURGUET, DAVID LO JACONO, IMFT - CNRS/INP/UPS — The flow-induced vibrations of a circular cylinder, free to oscillate in the cross-flow direction and subjected to a forced rotation about its axis, are studied by means of two- and three-dimensional numerical simulations, at a Reynolds number equal to 100. This problem serves as a paradigm to investigate the impact of symmetry breaking on the phenomenon of vortex-induced vibrations (VIV), previously described in the non-rotating case. The cylinder exhibits free oscillations up to a rotation rate close to 4. Under forced rotation, the vibration amplitude reaches 1.9 diameters, i.e. three times the maximum amplitude in the non-rotating case. Contrary to galloping responses, the free vibrations of the rotating cylinder are found to involve a condition of wake-body synchronization similar to the lock-in condition driving non-rotating cylinder VIV. A variety of flow patterns including novel asymmetric wake topologies is identified; it is shown that free oscillations may develop in the absence of vortex shedding. The symmetry breaking substantially alters the fluid force spectra and phasing mechanisms. The flow three-dimensional transition is found to occur at high rotation rates; its influence on the fluid-structure system behavior is analyzed.

4:40PM L12.00006 Vortex-Induced Vibrations of a Flexibly-Mounted Cyber-Physical Rectangular Plate[1] , KYOHEI ONOUE, BENJAMIN STROM, ARNOLD SONG, KENNETH BREUER, Brown University — We have developed a cyber-physical system to explore the vortex-induced vibration (VIV) behavior of a flat plate mounted on a virtual spring damper support. The plate is allowed to oscillate about its mid-chord and the measured angular position, velocity, and torque are used as inputs to a feedback control system that provides a restoring torque and can simulate a wide range of structural dynamic behavior. A series of experiments were carried out using different sized plates, and over a range of freestream velocities, equilibrium angles of attack, and simulated stiffness and damping. We observe a synchronization phenomenon over a wide range of parameter space, wherein the plate oscillates at moderate to large amplitude with a frequency dictated by the natural structural frequency of the system. Additionally, the existence of bistable states is reflected in the hysteretic response of the system. The cyber-physical damping extracts energy from the flow and the efficiency of this harvesting mechanism is characterized over a range of dimensionless stiffness and damping parameters.

[1]This research is funded by the Air Force Office of Scientific Research (AFOSR)

4:53PM L12.00007 Vortex-induced vibration of a curved cylinder , BANAFSHEH SEYED-AGHAZADEH, COLLIN BUDZ, YAHYA MODARRES-SADEGHI, University of Massachusetts, Amherst — Vortex-induced vibration of a curved circular cylinder free to oscillate in the crossflow direction is studied experimentally. Both concave and convex orientations (with respect to the oncoming flow direction) were considered. The system had a mass ratio of 3.6 and a structural damping of 0.01. The amplitudes and frequencies of oscillations were measured in a Reynolds number range of Re=500-2700. It was found that the amplitude of oscillations in both configurations was decreased compared with a vertical cylinder with the same mass ratio. The lock-in range was also wider in both cases compared with the lock-in range of a vertical cylinder. Higher harmonic components in the crossflow force were observed in both cases. In the entire lock-in range, the crossflow displacement and force stayed in phase, however, the contribution of the higher harmonic force components became more significant at higher reduced velocities. Dye flow visualizations showed that the vortices were shed in parallel to the curved cylinder, when the cylinder was free to oscillate.

5:06PM L12.00008 Suppression of Vortex Induced Vibrations by Fairings , YUE YU, Brown University, HONGMEI YAN, YIANNIS CONSTANTINIDES, OWEN OAKLEY, Chevron Energy Technology Company, GEORGE KARNIADAKIS, Brown University — Fairings are nearly-neutrally buoyant devices, which are fitted along the axis of long circular risers to suppress vortex induced vibrations (VIV) and possibly reduce the drag force. Here we study numerically how VIV can be practically eliminated by using free-to-rotate fairings. Since the mass ratio and rotational inertia are both low for the fairings, direct numerical simulations based on standard flow-structure interaction algorithms fail because of the so-called added mass effect. To resolve this problem we introduce fictitious methods and successfully stabilize the simulations. In particular, we investigate the effect of rotational friction C_f on the stabilization effect of the fairings. We found that there exists a critical value for the rotational friction, and when C_f is close to this value, large oscillations and unsymmetrical trajectories can be observed for the riser but for smaller C_f values VIV are suppressed substantially.

5:19PM L12.00009 Flapping dynamics of an inverted flag[1] , DAEGYOUM KIM, JULIA COSSE, MORTEZA GHARIB, California Institute of Technology — The dynamics of an inverted flag are investigated experimentally in order to find the conditions under which flapping can occur. In contrast to a general flag with a fixed leading edge and a free trailing edge, the inverted flag of our study has a free leading edge and a clamped trailing edge. The behavior of the inverted flag can be classified into three regimes based on its bending stiffness. Two quasi-steady regimes, straight mode and fully deflected mode, are observed, and limit-cycle flapping mode with large amplitude appears between the two quasi-steady regimes. Bistable states are found in both straight to flapping mode transition and flapping to deflected mode transition. The effect of mass ratio, relative magnitude of flag inertia and fluid inertia, on the bending stiffness range for flapping is negligible unlike the instability of the general flag. Because of unsteady fluid force, a flapping sheet can produce several times larger elastic strain energy than a sheet of the deformed mode, improving the conversion of fluid kinetic energy to elastic strain energy. According to the analysis of leading-edge vortex formation process, the time scale of optimal vortex formation correlates with efficient conversion to elastic strain energy during bending.

[1]This research is supported by the Gordon and Betty Moore Foundation.

Monday, November 25, 2013 3:35PM - 5:45PM –

Session L13 Granular Flows III: Jamming, Cooling and Force Transmission 301 - Shankar Subramaniam, Iowa State University

3:35PM L13.00001 Relaxation Time Scales for Dense Granular Systems , SHAOLIN MAO, Mechanical Engineering Department, University of Texas at El Paso — In study of shock waves, it is a common practice to assume that the thickness of a shock wave in thin, and that the equation of state of the material is applicable before and after the shock. While these assumptions are correct on gases with simple molecules. These assumptions need to be reexamined for granular systems, especially for dense granular systems, because the time scale to reestablish a steady or equilibrium state after an external perturbation could be comparable to the time scale of the physical problem itself. We study the physical time scales of particulate systems by using discrete element method (DEM). First, a simple shear force is imposed to a system with periodic perturbation of energy to mimic the temperature field. The calculation of contact stress and the velocity fluctuation shows the stress relaxation mechanism and the process of the system recovery to its original state after an external perturbation. In this talk, we also discuss the relationship between the segregation of a system and the stress relaxation of the symmetric and asymmetric components.

3:48PM L13.00002 Earthquakes in the Laboratory: Continuum-granular coupling , DREW GELLER, SERGIY GERASHCHENKO, SCOTT BACKHAUS, ROBERT ECKE, Los Alamos National Laboratory — Earthquakes in nature feature large tectonic plate motion at large scales of 10-100 km and local properties of the earth on the scale of the rupture width, of the order of meters. Fault gouge generally fills the gap between the large slipping plates and may play an important role in the nature and dynamics of earthquake events. We have constructed a laboratory scale experiment that represents a similitude scale model of this general earthquake description. Two photo-elastic plates (50 cm $\times$ 25 cm $\times$ 1 cm) confine approximately 3000 bi-disperse nylon rods (diameters 0.12 and 0.16 cm, height 1 cm) in a gap of approximately 1 cm. The plates are held rigidly along their outer edges parallel to the gouge with one held fixed while the other edge is driven at constant speed over a range of about 5 cm. The local stresses exerted on the plates are measured using their photo-elastic response, the local relative motions of the plates, i.e., the local strains, are determined by the relative motion of small ball bearings attached to the top surface, and the configurations of the nylon rods are investigated using particle tracking. We report statistical analyses of data obtained from these experimental probes and compare with different avalanche models.

4:01PM L13.00003 Statics and Dynamics of Force Networks in Dense Particulate Systems[1] , LOU KONDIC, New Jersey Institute of Technology, MIROSLAV KRAMAR, ARNAUD GOULLET, KONSTANTIN MISCHAIKOW, Rutgers University — The talk will focus on the properties of force networks found in discrete element simulations of isotropically compressed particulate systems. We will discuss how these properties evolve as the system goes through the jamming transition, with particular focus on the influence of interparticle friction and polydispersity. Then, we will discuss new measures that can be used to quantify the temporal evolution of force networks, and discuss this evolution as the systems go through jamming. The implemented computational technique is based on persistence analysis that allows to consider global properties of force networks. This technique, that has been only very recently applied to particulate matter,[2] allows to extract significant new information, going much beyond separation into "strong" and "weak" force networks. The proposed approach describes the considered networks in a precise and tractable manner, allowing to identify novel features which could be difficult or impossible to describe using other approaches.

[1]Supported by NSF Grant No. DMS-0835611.
[2]Phys. Rev. E **87**, 0422207 (2013)

4:14PM L13.00004 Shear Jamming for Slippery Granular Particles , DONG WANG, JOSHUA DIJKSMAN, JIE REN[1], ROBERT BEHRINGER, Duke University — Shear Jamming of granular materials was first found for systems of frictional disks, with a static friction coefficients $\mu_s \simeq 0.6$. Jamming by shear is obtained by starting from a zero-stress state with a packing fraction $\phi_S \leq \phi \leq \phi_J$ between ϕ_J (isotropic jamming) and a lowest ϕ_S for shear jamming. This phenomenon is associated with strong anisotropy in stress and the contact network in the form of "force chains," which are stabilized and/or enhanced by the presence of friction. The issue that we address experimentally is how reducing friction affects shear jamming. We use either Teflon disks, or disks that have been wrapped with Teflon, lowering the friction coefficient substantially from previous experiments. The Teflon disks were placed in a 2D shear apparatus (Ren et al., PRL 110, 018302 (2013)), with two rows of uncoated photoelastic particles at the periphery. The interior Teflon particles formed the "system," and the outer ring of photoelastic particles provided force data. For Teflon disks, shear jamming was also observed, but the difference $\phi_J - \phi_S$ was smaller than for higher friction particles. Ongoing work is focused on studies using the Teflon-wrapped particles, which completely fill the apparatus.

[1]Currently at Merck & Co.

4:27PM L13.00005 Microscopic Order Parameter for Shear Anisotropy for Systems near Shear Jamming , JIE REN, JOSHUA DIJKSMAN, ROBERT BEHRINGER, Duke University — Sheared granular systems at packing fractions between $\phi_s \leq \phi \leq \phi_J$ can exist in states with zero and nonzero stress. A system, prepared in a stress-free states in this density range, upon being sheared exhibits first fragile, then shear jammed states, both having high stress and fabric anisotropy. The onset of shear jammed states resembles an order-disorder transition. In recent work, we showed that the order appears in a force space (Bi et al. to appear, PRL). Here, we identify an order parameter associated with individual particles, making it possible to construct correlations in physical space. We identify local (particle-scale) order with Γ, the deviatoric part of the force-moment tensor. This is a real symmetric, traceless matrix characterized by two coefficients, a and b, such that $\Gamma = aU_1 + bU_2$, and where U_1 is diagonal with elements ± 1, and U_2 has 0's on the diagonal, and 1 for the off-diagonal elements. The U_i's are orthogonal under an appropriate scalar product. Then, (a, b) provides a vector particle-scale order parameter. Γ is additive over all particles, and is analogous to the magnetization in a spin system. Also, particles with orthogonal shear stresses now correspond to anti-parallel vectors. We use this representation to investigate both the collective order of the system and also correlations. This talk presents analysis of experimental data that explore the properties of this new order parameter.

4:40PM L13.00006 Imaging Forces on Stressed Spheres , NICOLAS BRODU, JOSHUA DIJKSMAN, Duke University, HU ZHENG, Tongji University, ROBERT BEHRINGER, Duke University — We study the quasi-static deformation of three dimensional sphere packings over a range of compressive stresses. We perform experiments on slightly polydisperse, nearly frictionless soft hydrogel spheres in a modified tri-axial shear apparatus. We measure boundary stresses and access microstructural information by 3D imaging the entire packing. By resolving particle deformations via custom written image analysis software, we extract particle contacts and forces. We address whether sheared frictionless spheres display dilatancy pressure, we measure the non-linear force response of a disordered packing under compression and explore the plastic rearrangements inside cyclically sheared and compressed packings.

4:53PM L13.00007 Shear-rate Dependent Regime Transition in Homogeneously sheared systems of Frictionless Cohesive Granules[1] , ERIC MURPHY, SRIRAM SUNDARARAJAN, SHANKAR SUBRAMANIAM, Iowa State University — We study regime transition behavior in systems of cohesive micron-sized granular particles in the absence of friction via soft sphere discrete element (DEM) simulations. Previous studies[2,3] have identified a shear-rate dependent regime transition, from Bagnold to quasi-static scaling, occurring below jamming volume fractions. The transition of interest is well-described by theories for non-equilibrium phase transitions. Most notably, this regime transition is accompanied by the emergence of a diverging meso-scopic length-scale based on the formation of local contact networks indicative of clustering. We identify the relevant non-dimensional quantities, e.g. ratio of cohesive potential to granular kinetic temperature, which mark the location of the critical transition and show that the fabric tensor may serve as a promising order-parameter. The study of such simple systems has broad implications for the constitutive modeling of other athermal systems, and illuminates the growing need for the modeling of non-local effects in flows of macroscopic particles.

[1]We gratefully acknowledge the support for this work from NSF grant no. 0927660.
[2]Aarons, L. Sundaresan, S. Powder Tech. 169 (2006) 10–21.
[3]Rognon, P.G. et al. J. Fluid Mech. 596 (2008) 21–47.

5:06PM L13.00008 ABSTRACT WITHDRAWN –

5:19PM L13.00009 Mechanics of sequential jamming and unjamming phenomena in a multi-exit orifice silo[1] , AMIT KUNTE, ASHISH ORPE, PANKAJ DOSHI, Chemical Engineering Division, National Chemical Laboratory, Pune India 411008 — We have investigated the flow of a two dimensional granular assembly draining through a flat bottomed silo having multiple exit orifices using DEM simulations. The width of the central orifice of the silo is fixed at $3.5d$ which is small enough to cause jamming (or no-flow) through the orifice. Here d is the mean particle diameter. The width of the other nearby orifices is kept much more than $3.5d$, thus, ensuring continuous flow of particles through them. Interestingly, this continuous flow of particles in the vicinity interacts with the assembly of jammed particles above the central orifice causing rearrangements and ultimately unjamming the assembly leading to a smooth flow. During the entire drainage of the silo, the central orifice undergoes this sequence of jamming-unjamming event several times, the frequency of which depends on its proximity to the nearby orifices. We focus primarily on understanding this jamming-unjamming transition by investigating the contact force network and the normal force distributions. Our preliminary results show that the tails of the force distributions in the jammed region decay slower than those for the flowing regions. This qualitative behaviour is found to be independent of any prior rearrangement history.

[1]Department of Science and Technology, India, (Grant No. SR/S3/CE/037/2009)

5:32PM L13.00010 Shear jamming in granular materials[1] , JIE ZHANG, Shanghai Jiaotong University — For frictionless particles with purely repulsive interactions, there is a critical packing fraction ϕ_J below which no jammed states exist. Recent experiments have shown that applying shear to a stress-free initial state can generate states which are either fragile or shear jammed depending on the way the force-network is percolated (Bi et al Nature 2011). The nature of the jamming transition however is obscured because the existence of friction between the system and the third dimension. A new apparatus at SJTU has been designed to completely eliminate this friction by letting the particles float on the surface of a shallow water layer, which allows a study of the more detailed nature of the shear-jammed states and the transition from an unjammed state to a shear-jammed state. In this study, we also use high-precision force sensors to monitor the dynamical changes near the jamming transition. We further combine numerical simulations with the experiments to diagnose the nature of this jamming transition and its possible dependence on certain particle properties.

[1]The work at SJTU is in collaboration with Ling Zhang and Jie Zheng. The numerical simulations are in collaboration with Maobin Hu at Univ. of Sci. & Tech. of China.

Monday, November 25, 2013 3:35PM - 5:32PM –

Session L14 Experimental Techniques VI: Turbulence/Fluorescence 302 - David Dowling, University of Michigan

3:35PM L14.00001 A non-intrusive velocity measurement technique for naturally-occurring turbulent shear flows , CHIN HEI NG, RYAN KEEDY, ALBERTO ALISEDA, University of Washington — Turbulent shear flows are common in nature (atmospheric low level jets, ocean hydrothermal vents, volcanic eruptions, convective cells, etc) and play an important role in environmental processes. These frequently inaccessible and measurements that require a physical probe are both restricted and inaccurate. We propose a non-intrusive technique that estimates the velocity of these flows by taking advantage of the existence of natural markers, such as condensation droplets, gas bubbles, reflective turbulent features on the mixing interface, that can be observed in video of the phenomena. Displacement of these markers, assumed to behave as fluid material elements, is measured by digital image correlation, and that velocity is associated with the fluid superlayer at the mixing interface between the "seeded" and unseeded flows. The relationship between these superficial velocities and the velocity in the interior of the flow has been investigated in a turbulent round jet laboratory experiment that allows for variations in jet Reynolds number, jet density, and jet viscosity. Influence of imaging parameters was also studied, with particular relevance to the application of this technique to the field.

3:48PM L14.00002 PIV Measurements of Turbulent Flow Over a Permeable Wall using a Refractive-Index Matching Approach , T. KIM, G. BLOIS, J.L. BEST, K.T. CHRISTENSEN, Univ. of Illinois — Turbulent flows over permeable walls occur in a variety of natural environments and engineering applications. Unlike classical and widely-studied flows over impermeable walls, the peculiar dynamics of flow generated by permeable walls are poorly understood. Early studies suggest that the well-known higher energy dissipation induced by permeability (as compared to impermeable walls with similar roughness) can be explained by unveiling the flow interactions within the transition layer that forms at the interface between the overlying flow and the permeable wall. To overcome the challenges associated with quantifying the flow character both above and within a permeable wall, a Refractive-Index-Matching (RIM) approach was employed. Doing so facilitated optical access to the fluid flowing through the permeable wall, thus yielding direct PIV pore-space flow measurements within the transition layer. The permeable wall was formed by packing acrylic spheres in a cubic arrangement and was then immersed in an aqueous solution of sodium iodide at a concentration and temperature that ensured accurate refractive index match with the wall. Measurements were focused on the flow across the wall interface and the turbulent attributes of these surface–subsurface interactions were detailed.

4:01PM L14.00003 Simultaneous Velocity and Vorticity Measurement in Turbulence[1] , HUIXUAN WU[2], HAITAO XU, EBERHARD BODENSCHATZ, Max Planck Institute for Dynamics and Self-Organization — A new paradigm of simultaneous velocity and vorticity measurement is developed to study turbulence. Instead of deducing vorticity from velocities measured at neighboring points, this innovative approach detects the translations and rotations of micro-sized particles directly. These hydrogel particles are spherical, transparent, and encapsulate micro-mirrors. This method outstands conventional ones, e.g., hotwire arrays or PIV because its spatial resolution is much higher. It does not require a non-zero mean flow, and it can provide all three vorticity components, which is not available from planar PIV data. Its principle is to illuminate the mirror and utilize the variation of the reflection directions to deduce the local flow vorticity. Meanwhile, the particle position is recorded as in normal particle tracking. Therefore, the velocity and vorticity of a particle can be obtained simultaneously in Lagrangian framework. The authors have made benchmark experiments to evaluate this novel method in Taylor Couette flows. The results show that the instantaneous vorticity measurement is as accurate as 3%. We are now setting up a von Karman disk pair device to study the turbulent flow. This novel technique will provide unprecedented information of high Reynolds number turbulence

[1]The first author thanks the Alexander von Humboldt Foundation

[2]Huixuan Wu is currently a Humboldt postdoc research fellow in MPI DS

4:14PM L14.00004 Measurement of turbulent wall shear stress in air using micro-pillars , EBENEZER GNANAMANICKAM, University of Melbourne / Embry Riddle Aeronautical University, KEVIN KEVIN, JASON MONTY, NICHOLAS HUTCHINS, University of Melbourne — The measurement of unsteady wall shear stress in a turbulent boundary layer, especially when the working medium is air, has been a historically challenging problem in experimental fluid mechanics. Recently the micro-pillar shear stress sensor (MPS3) has shown promise in this regard. The MPS3 is an array of micro-pillar mounted on the wall of a model. These micro-pillars deflect an amount proportional to the drag force it experiences. This drag force is proportional to the wall shear stress. The micro-pillar tip deflection is thus tracked using high-speed imaging to yield the unsteady wall shear stress. Here, the MPS3 is used to carry out unsteady wall shear stress measurements in a fully developed channel flow. Both static and dynamic calibrations of the sensor are presented. The wall shear stress statistics obtained in the fully developed channel flow are compared with those obtained from Direct Numerical Simulations (DNS) to provide an assessment of the sensor capabilities. Exemplary measurements such as two-dimensional temporal distribution of the wall shear stress are presented to highlight the capabilities of the sensor.

4:27PM L14.00005 Measurements of laboratory turbulence with the 2d-Laser Cantilever Anemometer , JAROSLAW PUCZYLOWSKI, JOACHIM PEINKE, MICHAEL HOELLING, University of Oldenburg — A newly developed anemometer, the 2d-Laser Cantilever Anemometer, was used to measure the two-dimensional wind speed vector in laboratory-generated turbulence. The anemometer provides a temporal and spatial resolution comparable or even higher to those of commercial hot-wires and thus is an excellent alternative for high-resolution measurements. The 2d-Laser Cantilever Anemometer uses a previously unseen measurement technique in the range of anemometers. The principle is adopted from atomic force microscopes (AFM). A tiny micro-structured cantilever is brought into the airflow, where it experiences a drag force due to the moving fluid. The resulting deflection is measured using the laser pointer principle. Unlike the measuring principle of hot-wires this technique can be applied in challenging environments such as in liquids or very close to walls. Our comparing measurements with the 2d-Laser Cantilever Anemometer and an x-wire were carried out in the wake of rigid bodies and grids. The results show a great agreement with regards to the increment statistics on various scales, power spectra and turbulence intensity, thus proving the new anemometer.

4:40PM L14.00006 Development of Krypton Planar Laser-Induced Fluorescence for Supersonic Flow Environments[1] , ROSS BURNS, CHRIS COMBS, NOEL CLEMENS, The University of Texas at Austin — Experimental work is presented on the development of krypton planar laser-induced fluorescence as a tracer in supersonic flows. Fluorescent tracers commonly used in compressible flowfields, such as nitric oxide, acetone, and toluene, have notable disadvantages when used in specific flow conditions that can include tracer condensation, reactivity, and general toxicity. Krypton, a noble gas, is immune to these deleterious effects over a much broader range of conditions including combustion environments. For these studies, the $5p[3/2]_2 \leftarrow 4p^6\ {}^1S_0$ electronic transition of krypton, accessible via two-photon absorption, is excited using a tunable sum-frequency generation (SFG) system set at the peak of the atomic absorption line around 214.7 nm. Data is presented on the fluorescence lifetimes and collisional quenching cross-sections over a broad range of conditions for krypton-air mixtures. The technique is demonstrated in a Mach 3 hypermixing flowfield to showcase its utility in a complex compressible and turbulent flow environment.

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

4:53PM L14.00007 Non-contacting Measurement of Oil Film Thickness Between Loaded Metallic Gear Teeth , DANIEL B. COX, STEVEN L. CECCIO, DAVID R. DOWLING, Mechanical Engineering, University of Michigan, Ann Arbor MI, 48109 — The mechanical power transmission efficiency of gears is depends on the lubrication condition between gear teeth. While the lubrication levels can be generally predicted, an effective in-situ non-contacting measurement of oil film thicknesses between loaded metallic gear teeth has proved elusive. This study explores a novel oil film thickness measurement technique based on optical fluence, the light energy transmitted between loaded gear teeth. A gear testing apparatus that allowed independent control of gear rotation rate, load torque, and oil flow was designed and built. Film thickness measurements made with 5-inch-pitch-diameter 60-tooth spur gears ranged from 0.3 to 10.2 mil. These results are compared with film thickness measurements made in an earlier investigation (MacConochie and Cameron, 1960), as well as with predictions from two film thickness models: a simple two-dimensional squeezed oil film and the industry-accepted model as described by the American Gear Manufacturers Association (AGMA 925, 2003). In each case, the measured film thicknesses were larger than the predicted thicknesses, though these discrepancies might be attributed to the specifics the experiments and to challenges associated with calibrating the fluence measurements. [Sponsored by General Electric]

5:06PM L14.00008 Plasma Electron Density Measurements Using Phase-Sensitive FTIR Interferometry[1] , BRIAN NEISWANDER, ERIC MATLIS, THOMAS CORKE, University of Notre Dame — This work investigates the use of low-temperature plasma as an adaptive medium for high-bandwidth aero-optic wavefront control. To better understand plasma's optical properties (refractive index), a new diagnostic technique has been developed to simultaneously measure the average plasma electron density and heavy particle density. The technique uses phase-sensitive Fourier transform infrared (FTIR) interferometry to measure the optical dispersion of plasma across a spectrum of far-infrared wavelengths. The plasma electron density and heavy particle density values are determined using a least-squares analysis. This presentation describes the experimental setup and preliminary data from the measurement system.

[1]This work was supported by DARPA Award Number N65236-12-1-1001.

5:19PM L14.00009 Simultaneous measurement of flow over and transmigration through a cultured endothelial cell layer , LORI LAMBERT, University of Nebraska - Lincoln, IRAKLIS PIPINOS, TIMOTHY BAXTER, JASON MACTAGGART, University of Nebraska Medical Center, GEORGE KARNIADAKIS, Brown University, DEREK MOORMEIER, KENNETH BAYLES, University of Nebraska Medical Center, TIMOTHY WEI, University of Nebraska - Lincoln — This talk focuses on the methodologies associated with the integration of temporally and spatially resolved μPIV measurements of flow over live endothelial cells with measurements of chemical transport through and across the cells. The ultimate goal of the study is to examine and model the transport and transmigration of key agents responsible for the formation of atherosclerotic plaques. Flow over endothelial cells cultured in a microchannel was measured using μPIV. By making measurements in a number of planes parallel to the wall, key dynamic quantities such as shear and pressure distributions, along with surface topography could be computed from the flow measurements. Experiments were conducted in a 65 μm x 65 μm cross section microchannel at shear rates up to 20 dynes/cm^2. Changes in cell conformation as a function of time after flow was started were examined. The deposition and transmigration of LDL was also examined using fluorescent-tagged LDL molecules.

Monday, November 25, 2013 3:35PM - 5:58PM –

Session L16 Biofluids: Medical Devices 304 - Kerem Pekkan, Carnegie Mellon University

3:35PM L16.00001 Computational Simulations of Inferior Vena Cava (IVC) Filter Placement and Hemodynamics in Patient-Specific Geometries , KENNETH AYCOCK, Penn State University, SHANKAR SASTRY, University of Utah, JIBUM KIM, Penn State University, SUZANNE SHONTZ, Mississippi State University, ROBERT CAMPBELL, KEEFE MANNING, Penn State University, FRANK LYNCH, Penn State Hershey Medical Center, BRENT CRAVEN, Penn State University — A computational methodology for simulating inferior vena cava (IVC) filter placement and IVC hemodynamics was developed and tested on two patient-specific IVC geometries: a left-sided IVC, and an IVC with a retroaortic left renal vein. Virtual IVC filter placement was performed with finite element analysis (FEA) using non-linear material models and contact modeling, yielding maximum vein displacements of approximately 10% of the IVC diameters. Blood flow was then simulated using computational fluid dynamics (CFD) with four cases for each patient IVC: 1) an IVC only, 2) an IVC with a placed filter, 3) an IVC with a placed filter and a model embolus, all at resting flow conditions, and 4) an IVC with a placed filter and a model embolus at exercise flow conditions. Significant hemodynamic differences were observed between the two patient IVCs, with the development of a right-sided jet (all cases) and a larger stagnation region (cases 3-4) in the left-sided IVC. These results support further investigation of the effects of IVC filter placement on a patient-specific basis.

3:48PM L16.00002 Hemodynamics of Central Venous Catheters: experiments and simulations , MICHAEL BARBOUR, PATRICK MCGAH, ALICIA CLARK, CHIN HEI NG, University of Washington, KENNETH GOW, Seattle Children's Hospital, ALBERTO ALISEDA, University of Washington — Central venous catheters (CVC) are used to provide vascular access during hemodialysis in patients with end-stage kidney disease. Despite several advantages and widespread use, CVCs have a high incidence rate of clot formation during the interdialytic phase (48 hrs). In an attempt the prevent clot formation, hospitals routinely administer heparin, an anticoagulant, into the catheter after a dialysis session. It has been reported, however, that up to 40% of the heparin solution will leak into the blood stream during the interdialytic phase, placing the patient at risk for systemic bleeding incidences. The aim of this study is to determine the role that advective-diffusive transport plays in the heparin leaking process. Numerical simulations of heparin convective mass transfer have been conducted, showing that while advective losses may be significant at the tip, previous studies may be overestimating the total amount of heparin leakage. To validate the quantitative prediction from the simulations, P.L.I.F. is used to experimentally measure heparin transport from CVCs placed in an idealized Superior Vena Cava with physically accurate pulsatile flow conditions. Improved understanding of flow near the catheter tip is applied to improve catheter design and heparin locking procedures.

4:01PM L16.00003 A Novel Thin Film Nitinol Covered Neurovascular Stent Significantly Decreases Intra-Aneurysmal Flow In Vitro , YOUNGJAE CHUN, University of Pittsburgh, SOOJUNG HUR, Harvard University, MAHDIS SHAYAN, University of Pittsburgh, COLIN KEALEY, Neurosigma, DANIEL LEVI, UCLA School of Medicine, KP MOHANCHANDRA, DINO DI CARLO, GREGORY CARMAN, University of California, Los Angeles — A novel thin film nitinol (TFN) stent has been developed to promote aneurysm quiescence by diminishing flow across the aneurysm's neck. Laboratory aneurysm models were used to assess the flow changes produced by stents covered with different patterns of TFN. Flow diversion stents were constructed by covering Wingspan stents (Boston Scientific, DxL:4x20mm) with TFNs (i.e., 77 and 82 percent porosity). The flow changes that occur after deployment of two different porous TFN covered stent in intracranial aneurysm models were evaluated in vitro. The 82 percent porous TFN covered stent reduced the intra-aneurysmal mean flow velocity by 86.42 percent, while a 77 percent porous TFN covered stent reduced to intra-aneurysmal mean flow velocity to 93.44 percent compared to a nonstented model. Local wall shear rates were also significantly reduced in wide-neck aneurysm model (i.e., 97.52 - 98.92 percent) with TFN stent placement. The results showed that TFN covered stents significantly reduced intra-aneurysmal flow velocity magnitudes and local wall shear rates. This suggests that TFN covered stents with both 77 and 82 percent porosity have great potential to promote thrombosis in both wide-necked and fusiform aneurysm sacs.

4:14PM L16.00004 Validation of an open-source framework for the simulation of blood flow in biomedical devices[1] , ANNALISA QUAINI, University of Houston, TIZIANO PASSERINI, Emory University, UMBERTO VILLA, Lawrence Livermore National Laboratory, ALESSANDRO VENEZIANI, Emory University, SUNCICA CANIC, University of Houston — We discuss the validation of an open source framework for the solution of problems arising in hemodynamics. The framework is assessed through experimental data for fluid flow in an idealized medical device with rigid boundaries. The core of the framework is an open source parallel finite element library that features several algorithms for fluid problems. The numerical results for the flow in the idealized medical device are in good quantitative agreement with the measured axial components of the velocity and pressures for flow rates corresponding to laminar, transitional, and turbulent regimes. A detailed account of the methods is provided.

[1]Support through grants NSF DMS-1109189 and NIH R01 HL70531 is gratefully acknowledged.

4:27PM L16.00005 Numerical simulations of the hemodynamics impact of stent-malapposition in a circular idealized coronary artery , ERIC POON, ANDREW OOI, WEI PAN, YUN LIU, YUFEI YE, YUAN XUE, University of Melbourne, PETER BARLIS, Northern Health, Australia, STEPHEN MOORE, VLSCI, Australia — Pulsatile flow past two circular cylinder rings in tandem inside a circular pipe is carried out numerically at resting blood flow rate (around 200mL/min) to study the effect of stent-malapposition (distance between cylinders surface and the circular pipe wall) on the hemodynamics impact inside a coronary artery. The corresponding Reynolds number based on pipe diameter for this blood flow rate is $Re = 600$. Stent-malappostion is chosen to be 0.25–1 diameter from the circular pipe wall and the two circular cylinders are 36 diameters apart. At 0.25 diameter stent-malapposition, the flow between the cylinders and the wall slows down significantly as the boundary layers from the cylinder and the wall meet. At 0.5 diameter stent-malapposition, the flow between the leading cylinder and the wall increases substantially, leading to unsteady vortices rolling away from the wall and a dramatic increase in wall shear stress. However, the vortices behind the trailing cylinder are stable even though the two cylinders in tandem are 36 diameters apart as flow pusatility affects the velocity recovery behind the leading cylinder. At 1 diameter stent-malapposition, the vortices behind the leading cylinder become stable again.

4:40PM L16.00006 Measurements of flow past a bileaflet mechanical heart valve[1] , LAURA HAYA, STAVROS TAVOULARIS, University of Ottawa — A bileaflet mechanical heart valve has been inserted in an axisymmetric model of the aorta within a mock circulation apparatus with physiological pressure and flow variations. The velocity field behind the valve has been measured with laser Doppler velocimetry and particle image velocimetry. The results closely match those reported by similar studies. A triple jet emanated from the valve's orifices and regions of reverse flow formed in the sinus region. Velocity fluctuations were greatest in the shear layers of the jets. The average r.m.s. streamwise velocity fluctuation over the turbulent period was 0.22 m/s; its maximum value was 0.53 m/s and occurred at the onset of deceleration. Measurements with the valve inserted in an anatomical model of the aorta are planned for the near future. The present and future measurements will be compared to determine the effects of the aorta anatomy on the characteristics of flow through bileaflet valves. In particular, measurements of the viscous and turbulent shear stresses will be analyzed to identify possible locations of blood element damage, and regions of recirculation and stagnation will be identified as locations favourable to thrombus growth. The effects of flows in branching arteries and valve orientation will also be investigated.

[1]Supported by NSERC

4:53PM L16.00007 On the open/close performance of prosthetic heart valves at high frequencies , A. BELTRAN, R. ZENIT, Universidad Nacional Autonoma de Mexico — We report experimental observations of the performance of mechanical and biological prosthetic heart valves. The valves are mounted in a test circular channel conected to a flow system that emulates accelerated human-like conditions. The flow is generated by a high frequencie pulsative pump (in the range of 7 to 18 Hz). The objective of the investigation is to find the treshold conditions for which the open/close performance fails. Preliminary results show that for the mechanical valve the failure starts at 436 pulses/min, while for the biological valve, it starts a failing performance is observed for frequencies higher that 462 pulses/min. Even though these values are far from the heart rate in the human body, we use these measurements to further understand the structure-fluid interaction mechanics of the flow through heart valves.

5:06PM L16.00008 ABSTRACT WITHDRAWN –

5:19PM L16.00009 Effects of Pannus Formation on the Flow around a Bileaflet Mechanical Heart Valve[1] , WOOJIN KIM, HAECHEON CHOI, Seoul National University, JIHOON KWEON, DONG HYUN YANG, NAMKUG KIM, YOUNG-HAK KIM, Asan medical center/University of Ulsan, College of Medicine — A pannus, an abnormal layer of fibrovascular tissue observed on a bileaflet mechanical heart valve (BMHV), induces dysfunctions of BMHV such as the time delay and incomplete valve closing. We numerically simulate the flows around an intra-annular type BMHV model with and without pannus formation, respectively, and investigate the flow and bileaflet-movement modifications due to the pannus formation. Simulations are conducted at a physiological condition (mean flow rate of 5 l/min, cycle duration of 866 ms, and the Reynolds number of 7200 based on the inflow peak bulk velocity and inflow diameter). We model the pannus as an annulus with fixed outer radius and vary the inner radius of the pannus. Our preliminary results indicate that the flow field changes significantly and the bileaflet does not close properly due to the pannus formation. The detailed results will be given at the final presentation.

[1]Supported by the NRF Programs (NRF-2011-0028032, NRF-2012M2A8A4055647)

5:32PM L16.00010 The role of intraventricular vortices in the left ventricular filling?[1] , PABLO MARTINEZ-LEGAZPI, University of California San Diego, JAVIER BERMEJO, YOLANDA BENITO, MARTA ALHAMA, RAQUEL YOTTI, CANDELAS PEREZ DEL VILLAR, ANA GONZALEZ-MANSILLA, ALICIA BARRIO, FRANCISCO FERNANDEZ-AVILES, JUAN CARLOS DEL ALAMO, Hospital General Universitario Gregorio Maranon — The generation of vortices during early filling is a salient feature of left ventricular hemodynamics. Existing clinical data suggest that these intraventricular vortices may facilitate pulling flow from the left atrium. To test this hypothesis, we have quantitatively dissected the contribution of the vortex to intraventricular pressure gradients by isolating its induced flow in ultrasound–derived data in 20 patients with non-ischemic dilated cardiomyopathy (NIDCM), 20 age-matched healthy controls and 20 patients with hypertrophied cardiomyopathy. We have observed that, in patients with NIDCM, the hemodynamic forces were shown to be partially supported by the flow inertia whereas that effect was minimized in healthy hearts. In patients with hypertrophied cardiomiopathy such effect was not observed.

[1]Supported by grants, PIS09/02603, RD06/0010 (RECAVA), CM12/00273 (to CPV) and BA11/00067 (to JB) from the Instituto de Salud Carlos III, Spain. PML and JCA were partially supported by NIH grant 1R21 HL108268-01

5:45PM L16.00011 Is aspect ratio sufficient to classify intra-aneurysmal hemodynamics- a parametric approach , MICHAEL DURKA, ANNE ROBERTSON, University of Pittsburgh — Intracranial aneurysms are a vascular pathology in which a localized bulge is formed in the arterial wall, most often in a saccular shape. It is believed that the blood flow field within the aneurysm plays a critical role in the degradation of the wall. Aneurysm rupture has a high mortality risk. Since only a small fracture of aneurysms rupture, and common treatments have their own risks, it is desirable to identify a useful means of assessing rupture risk. Therefore, numerous groups have endeavored to identify a correlation between rupture risk and sac geometry or flow dynamics. However, no clinically useful parameters have been identified to date. Prior work has suggested that the aspect ratio (sac height/neck) could be useful for risk stratification due to its influence on the sac hemodynamics. In this work, we make of a previously developed parametric model of the aneurysm geometry to evaluate the influence of aspect ratio (sac height/sac neck) on flow dynamics, using computational fluid dynamics. In particular, we assess the influence of aspect ratio on the number of vortices in the aneurysm sac over a wide range of sac geometries. The conclusions obtained for the parametric model are then assessed in 20 clinical cases.

Monday, November 25, 2013 3:35PM - 5:45PM –

Session L17 Biofluids: Locomotion VII - Active Suspensions and Bacteria 305 - Sung Kwon Cho, University of Pittsburgh

3:35PM L17.00001 Marangoni-driven chemotaxis, chemotactic collapse, and the Keller-Segel equation , MICHAEL SHELLEY, HASSAN MASOUD, Applied Math Lab, Courant Institute, NYU — Almost by definition, chemotaxis involves the biased motion of *motile* particles along gradients of a chemical concentration field. Perhaps the most famous model for collective chemotaxis in mathematical biology is the Keller-Segel model, conceived to describe collective aggregation of slime mold colonies in response to an intrinsically produced, and diffusing, chemo-attractant. Heavily studied, particularly in 2D where the system is "super-critical", it has been proved that the KS model can develop finite-time singularities – so-called chemotactic collapse – of delta-function type. Here, we study the collective dynamics of *immotile* particles bound to a 2D interface above a 3D fluid. These particles are chemically active and produce a diffusing field that creates surface-tension gradients along the surface. The resultant Marangoni stresses create flows that carry the particles, possibly concentrating them. Remarkably, we show that this system involving 3D diffusion and fluid dynamics, exactly yields the 2D Keller-Segel model for the surface-flow of active particles. We discuss the consequences of collapse on the 3D fluid dynamics, and generalizations of the fluid-dynamical model.

3:48PM L17.00002 Coherent structures, globally aligned states, and hydrodynamic traffic jams in confined active suspensions , DAVID SAINTILLAN, ADRIEN LEFAUVE, Department of Mechanical Science and Engineering, University of Illinois at Urbana-Champaign — Strongly confined active liquids are subject to unique hydrodynamic interactions due to momentum screening and lubricated friction by the confining walls. Using numerical simulations based on a minimal model for swimmer dynamics and interactions, we demonstrate that two-dimensional dilute suspensions of fore-aft asymmetric polar swimmers in a Hele-Shaw geometry can exhibit a rich variety of collective behaviors depending on particle shape and density, including: coherent polarized density waves with global alignment, stationary aster-shaped clusters, persistent counter-rotating vortices, density shocks and rarefaction waves. We also substantiate these various phenomena using a linear stability analysis and a nonlinear traffic flow model, both derived from a mean-field kinetic theory.

4:01PM L17.00003 A nonlocal kinetic theory for active suspensions in confined geometries , BARATH EZHILAN, DAVID SAINTILLAN, Department of Mechanical Science and Engineering, University of Illinois at Urbana-Champaign, Urbana, Illinois 61801, USA — We consider a suspension of biologically active particles confined by bounding walls separated by distances that are of the same-order as (or one order of magnitude higher than) the individual particle length. In such systems, the fluid velocity, particle concentration and orientation distribution vary on length scales that are comparable to that of the suspended active particles and a nonlocal theory is required to explain the dynamics. The theory presented here is based on our previous kinetic model for active suspensions, where nonlocal effects are taken into account by extending previous theories for passive fiber suspensions [Schiek and Shaqfeh (1995)]. Contacts between the active particles and the solid walls create sterically-excluded regions of particle configurations within a distance of a half particle length from the walls, and a rigorous no-flux boundary condition is imposed on the hypersurfaces separating the allowed and forbidden configurations. A pressure-driven flow is also imposed on this system and a numerical solution is utilized to study the concentration, orientation distributions, stress profiles, and effective viscosity. Comparisons are made to recent rheological measurements in confined bacterial suspensions [Gachelin et al (2013)].

4:14PM L17.00004 Rotors in low Re fluid: interactions and dynamics near a wall , ENKELEIDA LUSHI, PETIA VLAHOVSKA, Brown University — Active suspensions exhibit many interesting phenomena, e.g., self-organization and pattern formation. While collections of swimmers, which translate, have been extensively studied, rotors have received limited attention. We present a minimal model and numerical method to study the behaviour of externally or internally driven rotors in low Reynolds number flow. The rotors interact with each-other via the fluid as well as via excluded volumes. We discuss the coupled dynamics of two or more such particles, and their behaviour near a wall. Two same-spin rotors oscillate about their own center of mass with the oscillation time-scale depending on the particle aspect ratio, while their slow dynamics describes a large circular trajectory about the pair's center of mass. Two opposite-spin rotors perform, on average, co-operative self-propulsion in the direction perpendicular to their separation as well as oscillate about their centres of mass. A single rotor can move along a wall as it performs a co-operative self-propulsion with its own image. Last, we discuss the coupled dynamics and trajectories of many rotors.

4:27PM L17.00005 Active clusters and swimming crystals: instabilities and nonlinear dynamics in aggregates of model microswimmers , ARTHUR EVANS, University of Massachusetts, Amherst — Self-propelled particles, from synthetic Janus swimmers to living microorganisms, behave very differently when they are in isolation, near boundaries, or in the presence of their fellow swimmers. Although many systems studied involve dilute suspensions of these active particles, for large volume fractions near-field fluid mechanics and boundary effects can dominate the dynamics. In this talk I will use the "squirmer" model for self-propelled microswimmers to discuss the nonlinear dynamics of spherical particles that are nearly touching; in this limit the fluid mechanics are vastly simplified and predictions can be made for dynamical self-assembly and the overall motion of aggregates. For small clusters the behavior is analytically tractable, and results for the stability of paired swimmers can be recovered, while for aggregates of three or more particles chaotic behavior is predicted. In the limit of large numbers of nearly close-packed particles the lubrication analysis presented here can be used to predict instabilities in active colloidal crystals.

4:40PM L17.00006 Orientational order in two-dimensional confined active suspensions , ALAN CHENG HOU TSANG, EVA KANSO, University of Southern California — Geometric confinement in physical space is important for the studies of the collective motion of active suspensions. The reasons are two-fold: motile biological micro-organisms or active collides are always subject to different types of confinement in their swimming environment; The existence of confinement can significantly affects hydrodynamic interactions between the swimmers and thus changes the nature of collective motion. We focus on the situation when the swimmers are confined between two parallel plates such that the motion of the particles are restricted to two dimensions. In this case, the far-field hydrodynamic effect of a swimmer is no longer given by a force-dipole, which has been used in numerous studies on discrete numerical simulations and continuum theories. Instead, the far-field effect of a confined swimmer is given by a potential-dipole. Using a potential-dipole model in doubly-periodic domain, we perform numerical simulations to probe into the collective dynamics of confined active suspensions. We show that isotropic suspensions of swimmers are unstable and develop long time polar orientation order. This results in coherent clusters swimming in the same direction, reminiscent to the collective behavior usually observed in phenomenological models.

4:53PM L17.00007 Effects of bubble length and excitation frequency on micro propulsion by oscillating bubble[1] , JIAN FENG, SUNG KWON CHO, University of Pittsburgh — Previously, we have showed that an oscillating micro bubble column trapped in a one-end open channel can generate a propulsion force in the presence of an acoustic excitation [1]. The main mechanism for this propulsion is generation of asymmetric flows within the cyclic period of excitation. In particular, the amplitude of the bubble interface oscillation at the open end of the channel seems to be highly correlated to the propulsion strength. In addition, the oscillation amplitude highly depends on the excitation frequency as well as the bubble length. This means that the frequency and bubble length can be key parameters for controlling the propulsion strength. In this talk, we discuss how the bubble length and excitation frequency affect the micro propulsion. As the bubble length and the excitation frequency are varied, the oscillation amplitude, the strength of generated flows near the oscillating bubble and the propulsion speed are measured. Based on the measurements, the relation of these parameters with the propulsion strength is investigated.

[1] Jian Feng and Sung Kwon Cho, MEMS2013 Conference, pp. 63-66.

[1]This work is supported by the NSF project (NSF ECCS 1029318).

5:06PM L17.00008 Stochastic dynamics of active Brownian spheres in linear flows , MARIO SANDOVAL, Universidad Autonoma Metropolitana, ERIC LAUGA, University of Cambridge — Most classical work on the hydrodynamics of low-Reynolds swimming addresses deterministic locomotion in quiescent environments. Thermal fluctuations in fluids are known to lead to a Brownian loss of the swimming direction and to effective long-time diffusion. As most swimming cells or synthetic swimmers are surrounded by external flows, we consider theoretically the stochastic dynamics of a model active particle (a self-propelled sphere) in a steady general linear flow. We derive a general formulation for all components of the long-time mean-square displacement tensor and apply our general results analytically to the case of a steadily-swimming particle in three different external linear flows (pure rotation, shear, and extension). Self-propulsion leads to the same long-time temporal scalings as for passive particles but with increased coefficients. By comparing the active terms with those obtained for passive particles we see that swimming can lead to enhancement of the mean-square displacements by orders of magnitude, and could be relevant for biological organisms or synthetic swimming devices in fluctuating environmental or biological flows.

5:19PM L17.00009 Swimming bacteria at complex interfaces , DIEGO LOPEZ[1], ERIC LAUGA[2], Dept. Mechanical and Aerospace Engineering, University of California San Diego, USA — Swimming microorganisms such as bacteria often move in confined geometries. Such confinement can be caused by the presence of solid boundaries, free surfaces, or liquid interfaces. It is well established that confinement affects significantly locomotion, generating additional forces and torques on the bacteria. In the presence of a solid boundary (imposing a no-slip condition), microorganisms using helical propulsion undergo circular motion (clockwise in the case of E. coli). Conversely, close to a free (no-shear) surface the circular motion is reversed. However, realistic interfaces are complex, and experimental results do not always agree with theoretical predictions. In this work, we show, using analytical modeling, how different complex interfaces affect a nearby bacterium and modify its swimming kinematics.

[1]IUSTI UMR 7343, Polytech Marseille, France

[2]Dept. Applied Mathematics and Theoretical Physics Centre for Mathematical Sciences, Wilberforce Road, Cambridge, CB3 0WA, United Kingdom

5:32PM L17.00010 Artificial Rheotaxis , JEREMIE PALACCI, CSMR, NYU, STEFANO SACANNA, Dpt of Chemistry, NYU, ANAIS ABRAMIAN, KASEY HANSON, DAVID PINE, PAUL CHAIKIN, CSMR, NYU, CSMR, NYU TEAM — Self propelled colloids realize a controlled realization of an artificial bacterium. However living systems present a range of advanced properties such as the migration in gradients, or taxis, based on complex conformational change of proteins. For example, rheotaxis, the directed movement of an organism resulting from a fluid flow, has been reported notably for fish, e.g. salmon, or spermatozoa. Here, we present experimental observations of artificial rheotaxis, i.e. upstream migration of self propelled particles in the presence of a flow. We will present a simple model to account for this surprising effect. In the absence of biological component, this effect is intriguing and questions the ingredients at stake in the living matter.

Monday, November 25, 2013 3:35PM - 5:06PM –

Session L18 Biofluids: General V 306/307 - Jiarong Hong, University of Minnesota

3:35PM L18.00001 The Hydrodynamics and Odorant Transport Phenomena of Olfaction in the Hammerhead Shark , ALEX RYGG, BRENT CRAVEN, Penn State University — The hammerhead shark possesses a unique head morphology that is thought to facilitate enhanced olfactory performance. The olfactory organs, located at the distal ends of the cephalofoil, contain numerous lamellae that increase the surface area for olfaction. Functionally, for the shark to detect chemical stimuli, water-borne odors must reach the olfactory sensory epithelium that lines these lamellae. Thus, odorant transport from the aquatic environment to the sensory epithelium is the first critical step in olfaction. Here we investigate the hydrodynamics and odorant transport phenomena of olfaction in the hammerhead shark based on an anatomically-accurate reconstruction of the head and olfactory chamber from high-resolution micro-CT and MRI scans of a cadaver specimen. Computational fluid dynamics (CFD) simulations of water flow in the reconstructed model reveal the external and internal hydrodynamics of olfaction during swimming. Odorant transport in the olfactory organ is investigated using a multi-scale approach, whereby molecular dynamics (MD) simulations are used to calculate odorant partition coefficients that are subsequently utilized in macro-scale CFD simulations of odorant deposition. The hydrodynamic and odorant transport results are used to elucidate several important features of olfactory function in the hammerhead shark.

3:48PM L18.00002 An Experimental Study of Flow Separation Control by Shortfin Mako Shark Skin[1] , FARHANA AFROZ, AMY LANG, University of Alabama, PHILIP MOTTA, MARIA HABEGGER, University of South Florida — The shortfin mako shark (Isurus oxyrinchus) is a fast swimmer and has incredible turning agility. Shark skin is covered with flexible scales and this bristling capability may result in a unique Boundary Layer Control (BLC) method to reduce drag. It is hypothesized that scales bristle when the flow above it is reversed, and between the bristled scales embedded micro-vortices form in the cavities which induce boundary layer mixing and aid in delaying flow separation. To testify this hypothesis, samples of mako shark skin have been tested in a water tunnel under various strengths of adverse pressure gradient (APG). Laminar and turbulent separation over shark skin was studied experimentally using Time-Resolved Digital Particle Image Velocimetry (TR-DPIV) system, where the APG was generated and varied using a rotating cylinder. Then shark skin results were compared with that of a flat plate data for a given amount of APG. The study reveals that shark skin is capable of controlling both laminar and turbulent flow separation.

[1]Support under NSF grant 0932352 is gratefully acknowledged. First author Farhana Afroz was also supported by a scholarship through the Alabama EPSCoR Graduate Research Scholars Program.

4:01PM L18.00003 3D flow investigation near the denticles of biomimetic shark skin model using Digital In-line Holographic Microscopy[1] , MOSTAFA TOLOUI, JIARONG HONG, University of Minnesota — It has been hypothesised that the complex microscopic denticles on a shark skin reduce the total drag for a swimming shark. However, the fundamental mechanism of this hydrodynamic function is not fully understood due to the inability to reproduce the complex shark surface and resolve the detailed flow around the skin denticles. Here we report a preliminary experiment using a 3D printed transparent rough surface replicating the morphological features of real shark skin. The model skin consists of closely-packed denticles of 2 mm in scale, i.e. $\sim$ 10 times of the real size. Particle image velocimetry based on digital in-line holography is employed to measure 3D flow structures. To reduce optical abberration and enable imaging around the denticles, we use a fluid medium that has the same optical refractive index as that of the skin model. The experiment is conducted in 2"x2" square channel at a moderate Re number matching the general flow around a cruising shark. Several samples of the 3D velocity field amid and above the denticles are obtained. The follow-up research envisions a large dataset of these samples over the rigid/deformable model operated in stationary/undulating mode to ellucidate the dominant flow structures generated by the denticals.

[1]This research is collaborated with Prof. George Lauder's group.

4:14PM L18.00004 Shortfin Mako Skin: A Possible Passive Flow Control Mechanism for Drag Reduction[1] , JENNIFER WHEELUS, AMY LANG, MICHAEL BRADSHAW, The University of Alabama, PHILLIP MOTTA, MARIA HABEGGER, University of South Florida — The shortfin mako is one of the fastest and most agile ocean predators creating the need to minimize its pressure drag by controlling flow separation. One proposed method for flow control is the activation of small teeth-like denticles, on the order of 0.2 mm, that cover the skin of the shark. Biological studies of the shortfin mako skin have shown the passive bristling angle of their denticles to exceed 50 degrees in areas on the flank corresponding to the locations likely to experience separation first. It is proposed that reversing flow, as occurs at the onset of separation in a turbulent boundary layer, would activate denticle bristling and hinder local separation from leading to global separation over the shark. It has been shown on a biomimetic model that bristled denticles create cavities that support the formation of vortices that interact with the boundary layer. This interaction is thought to support momentum exchange and allow the flow to stay attached longer. This experiment focuses on the mechanism that triggers bristling of the real shark skin denticles and further explores the interaction those denticles foster with the boundary layer on a 3D biomimetic model using Digital Particle Image Velocimetry (DPIV).

[1]Support for this research by the NSF GRFP is gratefully acknowledged.

4:27PM L18.00005 Optimal lamellar arrangement in fish gills[1] , KEUNHWAN PARK, School of Mechanical and Aerospace Engineering Seoul National University, Korea, WONJUNG KIM, Department of Mechanical Engineering, Sogang University, Korea, HO-YOUNG KIM, School of Mechanical and Aerospace Engineering Seoul National University, Korea — We present the results of a combined theoretical and experimental investigation of the oxygen transport in fish gills. Efficient respiration is crucial to fish because of relatively low oxygen contents in water compared to that in air. Ordered structures of lamellae of fish gills offer extended surfaces for oxygen transport. While the more compact arrangement of the lamellae provides larger surface area for oxygen diffusion, it causes higher viscous resistance to water flow through the interlamellar space. This allows us to expect the optimal lamellar arrangement for maximizing the oxygen transport. By developing a dynamic model for oxygen transport in fish gills, we calculate optimal lamellar arrangement for maximizing oxygen transport. We demonstrate that the interlamellar distance of a broad range of fish species is consistent with the deduced optimal lamellar arrangement. Our results thus provide the first rationale for the relatively uniform interlamellar distance of many fish regardless of their size, appearance, and habitat.

[1]This work was supported by the Sogang Uinversity Research Grant of 2013 (201310009.01) and the National Research Foundation, Korea (2013034978)

4:40PM L18.00006 Viscous-elastic interaction as a mechanism to create adhesion in frogs' toe pads , AMIR GAT, ARIE TULCHINSKY, Technion - Israel Institute of Technology — The toe pads of frogs consist of soft hexagonal structures and a network of channels between and within the soft structures, containing a viscous liquid. It has been hypothesized that this configuration creates adhesion by allowing for long range capillary forces, or alternatively, that the channel network allows for exit of the viscous liquid and thus improve contact of the toe pad. In this work we suggest interaction between viscous flow and elastic forces as a mechanism to create temporary adhesion, even in the absence of capillary or van der Waals forces. We study the dynamics of a solid body covered with an array of protruding elastic cylinders, immersed within a viscous liquid, and pressed against a flat surface. Inertia is neglected and the elastic-viscous dynamics yield the governing differential equation describing the relative motion between the body and the surface. The compressed elastic cylinders apply a force acting to separate the solid body from the surface. The relative motion between the body and the surface creates a viscous flow and pressure field resisting the elastic force and significantly reducing the speed of separation. We show that the viscous-elastic interaction can prevent motion tangential and normal to the surface and can create temporary adhesion.

4:53PM L18.00007 The Hawaiian bobtail squid as a model system for selective particle capture in microfluidic systems. , JANNA NAWROTH, Caltech, MARGARET MCFALL-NGAI, University of Wisconsin at Madison, JOHN DABIRI, Caltech — Juvenile Hawaiian bobtail squids reliably capture and isolate a single species of bacteria, Vibrio fischeri, from inhaled coastal water containing a huge background of living and non-living particles of comparable size. Biochemical mechanisms orchestrate a chain of specific interactions as soon as V.fischeri attach to the squid's internal light organ. It remains unclear, however, how the bacteria carried by the squid's ventilation currents are initially attracted to the light organ's surface. Here we present preliminary experimental data showing how arrangement and coordination of the cilia covering the light organ create a 3D flow field that facilitates advection, sieving and selective retention of flow-borne particles. These studies may inspire novel microfluidic tools for detection and capture of specific cells and particles.

Monday, November 25, 2013 3:35PM - 4:40PM –

Session L19 Nanofluids I 310/311 - Sandip Ghosal, Northwestern University

3:35PM L19.00001 Overlimiting current through ion concentration polarization layer: Hydrodynamic convection effects[1] , INHEE CHO, SUNG JAE KIM, Seoul National University — In this presentation, we experimentally investigated an effect of the hydrodymanic convective flow on an ion transport through nanoporous membrane in a micro/nanofluidic system. The convective motion of ions in an ion concentration polarization zone was controlled by external hydrodynamic inflows adjacent to the nanoporous membrane. The ion depletion region (which is regarded as a high electrical resistance) is spatially confined to a triangular shape with an additional hydrodymanic convective flow, resulting in a significant alternation in classical liming current value. Furthermore, the extreme spatial confinement can completely eliminate the limiting current region at higher flow rate so that one can obtain high current value which turns to be high power efficiency. Therefore, this mechanism would be utilized as minimizing power consumption for various electrochemical membrane systems such as fuel-cell, electro-desalination system and nanofluidic preconcentrator, etc.

[1]This work is supported by Basic Science Research Program (2013R1A1A1008125) and Future-based Technology Development Program (Nano Fields) (2009-0082952) through the NRF funded by the Ministry of Science, ICT & Future Planning.

3:48PM L19.00002 Hydrodynamic flow in the vicinity of a nanopore in response to an applied voltage[1] , MAO MAO, Department of Mechanical Engineering, Northwestern University, SANDIP GHOSAL, Department of Mechanical Engineering and (by courtesy) Engineering Science & Applied Mathematics, Northwestern University, GUOHUI HU, Shanghai Institute of Applied Mathematics and Mechanics — Continuum simulation and analytical modeling is employed to study ion transport and fluid flow through a nanopore in a solid-state membrane under an applied voltage. The ion distribution near the surface of the membrane arises due to the combined effect of the intrinsic surface charge as well as concentration polarization due to the applied field. It gives rise to an electric pressure that drives hydrodynamic flow in the vicinity of the pore. There is a net hydrodynamic flow through the nanopore due to the asymmetry in the Debye layer induced by the membrane surface charge. The qualitative behavior is similar to that observed in a previous study using molecular dynamic simulations. The flow strength is a strongly nonlinear function of the applied field. Combination of electrophoretic and hydrodynamic effects can lead to ion selectivity in terms of valences and this could have some practical applications in separations.

[1]This work was supported by grant number R01HG004842 from the National Human Genome Research Institute, National Institutes of Health. One of us (SG) acknowledges support from the Leverhulme Trust (UK).

4:01PM L19.00003 Effect of chain stiffness on interfacial slip in nanoscale polymer films , NIKOLAI PRIEZJEV, Wright State University — The results obtained from molecular dynamics simulations of the friction at an interface between polymer melts and weakly attractive crystalline surfaces are reported. We consider a coarse-grained bead-spring model of linear chains with adjustable intrinsic stiffness. The structure and relaxation dynamics of polymer chains near interfaces are quantified by the radius of gyration and decay of the time autocorrelation function of the first normal mode. We found that the friction coefficient at small slip velocities exhibits a distinct maximum which appears due to shear-induced alignment of semiflexible chain segments in contact with solid walls. At large slip velocities, the friction coefficient is independent of the chain stiffness. The data for the friction coefficient and shear viscosity are used to elucidate main trends in the nonlinear shear rate dependence of the slip length. The influence of chain stiffness on the relationship between the friction coefficient and the structure factor in the first fluid layer is discussed. Financial support from the National Science Foundation (CBET-1033662) is gratefully acknowledged.

4:14PM L19.00004 Diffusion Monte Carlo ab initio calculations to study wetting properties of graphene , YANBIN WU, HUIHUO ZHENG, LUCAS WAGNER, N.R. ALURU, University of Illinois — For applications of graphene in water, including for example desalination and DNA sequencing, it is critical to understand the wetting properties of graphene. In this work, we investigate the wetting properties using data from highly accurate diffusion quantum Monte Carlo (DMC) calculations, which treat electron correlation explicitly. Our DMC data show a strong graphene-water interaction, indicating graphene surface is more hydrophilic than previously believed. This has been recently confirmed by experiments [Li et al. Nat. Mater. 2013, doi:10.1038/nmat3709]. The unusually strong interaction can be attributed to weak bonding formed between graphene and water. Besides its inadequate description of dispersion interactions as commonly reported in the literature, density function theory (DFT) fails to describe the correct charge transfer, which leads to an underestimate of graphene-water binding energy. Our DMC calculations can provide insight to experimentalists seeking to understand water-graphene interfaces and to theorists improving DFT for weakly bound systems.

4:27PM L19.00005 Prediction of the effective force on DNA in a nanopore based on density functional theory[1] , GUOHUI HU, WENYUE TANG, Shanghai Institute of Applied Mathematics and Mechanics, Shanghai University — We consider double-strand DNA voltage-driven translocation through a nanopore in the present study. By assuming the DNA is coaxial with the cylindrical nanopore, a hydrodynamic model for determining effective force on a single DNA molecule in a nanopore was presented, in which density functional theory (DFT) combined with the continuum Navier-Stokes (NS) equations is utilized to investigate electro-osmotic flow and the viscous drag force acting on the DNA inside a nanopore. Surface charge on the walls of the nanopore is also taken into account in our model. The consistence between our calculation and the previous experimental measurement indicates that the present theoretical model is an effective tool to predict the hydrodynamic resistance on DNA. Results show that charge inversion, which cannot be obtained by the Poisson-Boltzmann (PB) model, will reduce electro-osmotic velocity, or even lead to flow reversal for higher salt concentration. This is helpful to raise the effective force profoundly in the overscreening region.

[1]This work was supported by the National Science Foundation of China (Grant No. 11272197).

Monday, November 25, 2013 3:35PM - 5:45PM –

Session L20 Boundary Layers VI: Channel Flow and Flows over Superhydrophobic Walls 315 - R. Rubinstein, NASA Langley

3:35PM L20.00001 Representation of the velocity spectra and Reynolds stress co-spectrum in turbulent channel flow using resolvent modes[1] , RASHAD MOARREF, California Institute of Technology, ATI S. SHARMA, University of Southampton, JOEL A. TROPP, BEVERLEY J. MCKEON, California Institute of Technology — We represent the velocity field in channel flow as a weighted sum of a small number of 'resolvent modes' that are obtained by Fourier decomposition in the wall-parallel directions and time, and singular value decomposition of the resolvent operator in the wall-normal direction, following McKeon & Sharma (J. Fluid Mech., 2010). Building on previous efforts in which the Reynolds number scaling and geometric self-similarity of the resolvent modes were identified in a study of the streamwise velocity variance, we determine the resolvent mode weights required to minimize the deviation between an assembly of resolvent modes at Re_tau = 2003 and the time-averaged two-dimensional spectra (uu, vv, ww and uv) from direct numerical simulations (Hoyas & Jimenez, Phys. Fluids, 2006). While the spectra corresponding to small wavelengths can be approximated by a few resolvent modes, a larger number of modes is necessary for matching at large wavelengths. The Reynolds number scaling of the spectra and the associated implications of previously-identified self-similar attached eddies are further discussed.

[1]The support of Air Force Office of Scientific Research under grants FA 9550-09-1-0701 and FA 9550-12-1-0469 is gratefully acknowledged.

3:48PM L20.00002 High-Reynolds-number effects in turbulent channel flow: evidence from DNS , MATTEO BERNARDINI, SERGIO PIROZZOLI, PAOLO ORLANDI, University of Rome La Sapienza — The behavior of the incompressible turbulent channel flow is investigated through direct numerical simulation up to a Reynolds number ($Re_\tau \approx 4080$) at which phenomena typical of the asymptotic Reynolds number regime starts to be observed. Less than a decade of nearly-logarithmic variation is observed in the mean velocity profiles, with log-law constants $k \approx 0.386$, $C \approx 4.30$. A log layer is also observed in the spanwise velocity variance, as predicted by Townsend's attached eddy hypothesis, whereas the streamwise variance seems to exhibit a plateau, perhaps being still affected by low-Reynolds-number effects. Comparison with previous DNS data at lower Reynolds number suggests strong enhancement of the imprinting effect of outer-layer eddies onto the near-wall region. This mechanisms is shown to be associated with excess turbulence kinetic energy production in the outer layer, and it clearly reflects in flow visualizations and in the streamwise velocity spectra, which exhibit near-tonal behavior in the outer layer. Associated with the outer energy production site, we find evidence of a Kolmogorov-like inertial range, limited to the spanwise spectral density of u, whereas power laws with different exponents are found for the other spectra.

4:01PM L20.00003 Turbulent spots in a channel: an experimental study on the inner structure and the large-scale flow , JOSÉ EDUARDO WESFREID, GRÉGOIRE LEMOULT, PMMH (UMR 7636 CNRS-ESPCI), KONRAD GUMOWSKI, Warsaw University of Technology, Inst Aeronaut & Appl Mech, Poland, MINGYANG LUO, JEAN LUC AIDER, PMMH (UMR 7636 CNRS-ESPCI) — Transition to turbulence in plane Poiseuille flow in channels, occurs in presence of localized coherent structures, known as turbulent spots, composed of an assemblage of small-scale longitudinal vortices. We present an exhaustive experimental description of these spots, in a water channel of rectangular cross section. The test section's half height is h = 10mm, its length is 220h, and its width is 15h. We study the response of the flow to a single, short perturbation, which above Re =1300 will always trigger the development of a turbulent spot. The fine structure of the flow field inside and around a turbulent spot is obtained from the measurements of the three components of the velocity fields in a cross sectionnal plane with Time Resolved Stereoscopic Particule Image Velocimetry. The flow in the turbulent can be decomposed into a large-scale motion consisting of an asymmetric quadrupole centred on the spot and a small-scale part consisting of streamwise streaks. From the observations of the temporal evolution of the energy of the streamwise and spanwise velocity perturbations, it is suggested that a self-sustaining process can occur in a turbulent spot above a given Reynolds number. We also compare the dynamical evolution of the turbulent fluctuations and the mean flow distortions, with the predictions of reduced models predicting the main features of subcritical transition to turbulence.

4:14PM L20.00004 Temperature fluctuations in fully-developed turbulent channel flow with heated upper wall , CARLA BAHRI, MICHAEL MUELLER, MARCUS HULTMARK, Princeton University — The interactions and scaling differences between the velocity field and temperature field in a wall-bounded turbulent flow are investigated. In particular, a fully developed turbulent channel flow perturbed by a step change in the wall temperature is considered with a focus on the details of the developing thermal boundary layer. For this specific study, temperature acts as a passive scalar, having no dynamical effect on the flow. A combination of experimental investigation and direct numerical simulation (DNS) is presented. Velocity and temperature data are acquired with high accuracy where, the flow is allowed to reach a fully-developed state before encountering a heated upper wall at constant temperature. The experimental data is compared with DNS data where simulations of the same configuration are conducted.

4:27PM L20.00005 Reynolds Number Effects on Kinetic Energy Transfer from Outer Layer in Turbulent Channel Flows , YOSHINOBU YAMAMOTO, Yamanashi University, Japan, YOSHIYUKI TSUJI, Nagoya University, Japan — Kinetic energy transfer from the outer layer to the inner layer was investigated by means of DNS database of turbulent channel flows up to Re$\tau = 2000$. On assumption of Dean's empirical equation, energy transfer from the mean velocity component to the turbulence was increased in proportion of 1/7 power of turbulent Reynolds number, but the peak value of the turbulent production was saturated with Re$\tau = 6000$. This might be supported the importance of the second peaks of turbulent intensities in high-Re. In the view points of the ratio of kinetic energy transfer, both results of DNS database and theoretical analysis using logarithmic mean velocity profile represent that the boundary between the outer layer and the inner layer will be determined as the 20% of the channel half-length.

4:40PM L20.00006 An investigation of the flow modification in a turbulent channel with gain-based optimal forcing[1] , ARJUN SHARMA, University of Texas, Austin, RASHAD MOARREF, MITUL LUHAR, California Institute of Technology, Pasadena, DAVID GOLDSTEIN, University of Texas, Austin, BEVERLEY MCKEON, California Institute of Technology, Pasadena — Direct numerical simulations of turbulent channel at Re$_\tau$ $= 180$ with two-dimensional traveling wave forcing are performed. The chosen forcing parameters, that is, the wall-parallel wavelength and wave-speed, are representative of the near-wall cycle. The wall-normal forcing shape is obtained using the resolvent analysis of McKeon & Sharma (Journal of Fluid Mechanics, vol. 658, 2010). The results obtained from direct simulations are found to compare well with the resolvent analysis predictions for different forcing amplitudes. The turbulence statistics and visualizations of three-dimensional unsteady flow field shed light on the flow evolution in response to forcing.

[1]The support for this work was provided by AFOSR under grant number FA9550-12-1-0469 and the computational resources were provided by Texas Advanced Computing Center (TACC)

4:53PM L20.00007 Skin-friction Drag Reduction in Turbulent Channel Flow with Idealized Superhydrophobic Walls , AMIRREZA RATSEGARI, RAYHANEH AKHAVAN, The University of Michigan, Ann Arbor, MI 48109-2125 — Skin-friction drag reduction by super-hydrophobic (SH) surfaces was investigated using Lattice Boltzmann DNS in turbulent channel flow with SH longitudinal microgrooves on both walls. The liquid/gas interfaces in the SH microgrooves were modeled as flat, shear-free surfaces. Drag reductions (DR) ranging from 5% to 47% were observed for microgrooves of size $4 \leq g^{+0} = w^{+0} \leq 128$ in channels of bulk Reynolds number $Re_b = U_b h/\nu = 3600$ ($Re_{\tau_0} = u_{\tau_0} h/\nu \approx 230$), where g^{+0} and w^{+0} denote the widths of the slip and no-slip surfaces in base flow wall units. It is shown that in both laminar and turbulent flow, DR scales as $DR = U_s/U_b + \varepsilon$. In laminar flow, where DR is purely due to surface slip, $\varepsilon = 0$. In turbulent flow, ε remains negligible when the slip length is smaller than the thickness of the viscous sublayer. For $DR > 40\%$, where the effect of surface slip can be felt in the buffer layer, ε attains a small non-zero value. Analysis of turbulence statistics and turbulence kinetic energy budgets confirms that outside of a layer of size approximately one slip length from the walls, the turbulence dynamics proceeds as in regular channel flow with no-slip walls.

5:06PM L20.00008 A Chracterization of Superhydrophobic Surfaces for Skin-Friction Drag Reduction[1] , HYUNWOOK PARK, JOHN KIM, University of California, Los Angeles — A proper characterization of superhydrophobic surfaces (SHSs) was examined using direct numerical simulation of turbulent channel flows. A SHS was modeled through a shear-free boundary condition on the air-water interface. Within the considered Reynolds number range and SHS geometry, it was found that the drag reduction in turbulent channel flows was well correlated with the effective slip length normalized by viscous wall units. A maximum drag reduction was achieved when the effective slip length was on the order of 50 in viscous wall units. It was also shown that near-wall turbulence structures were significantly modified. Since the effective surface slip length can be interpreted as a depth of influence into which SHS affect the flow in the wall-normal direction, this result demonstrated that a SHS achieved its drag reduction by affecting those turbulence structures within the buffer layer of the turbulent channel flow. The present results also showed that the relative size between near-wall turbulence structures and the SHS geometry was an important parameter for drag reduction.

[1]Supported by ONR grant, N000141110503.

5:19PM L20.00009 Direct numerical simulation of turbulent flows over superhydrophobic surfaces: gas-liquid interface dynamics[1] , JONGMIN SEO, RICARDO GARCÍA-MAYORAL, ALI MANI, Stanford University — Superhydrophobic surfaces can induce large slip velocities for liquid flows, reducing the skin friction on walls, by entrapping gas pockets within the surface roughness. This work explores the onset mechanism leading to gas depletion through interface breakage under turbulent conditions. We conduct direct numerical simulations of flows over superhydrophobic walls. The superhydrophobic texture is conventionally modeled as a pattern of slip/no-slip boundary conditions for the wall-parallel velocities but, to take into account the dynamic deformation of the gas-liquid interface, we also introduce non-zero boundary conditions for the wall-normal velocity. These conditions are derived from the deformation of the interface in response to the overlying turbulent pressure fluctuations, following the Young-Laplace equation. Surface protrusions in the form of posts and streamwise-aligned ridges are studied, and results are presented as a function of the "deformability" of the gas-liquid interfaces, expressed as a Weber number. We will also discuss results for misaligned ridges.

[1]Supported by the Office of Naval Research and the Kwanjeong Educational Scholarship Foundation.

5:32PM L20.00010 Convective Air Mass Transfer in Submerged Superhydrophobic Surfaces: Turbulent Flow , CHRISTINA A. BARTH, HOOMAN VAHEDI TAFRESHI, MOHAMED GAD-EL-HAK, Department of Mechanical & Nuclear Engineering, Virginia Commonwealth University, Richmond, Virginia 23284-3015 — Longevity of entrapped air is an outstanding problem for using superhydrophobic coatings. Herein, we analyze from first principles a mass transfer problem. Using integral methods, we are able to extend our laminar flow solution presented last year to turbulent flows. We introduce an effective slip to the hydrodynamic boundary layer using a modified 1/7-power law velocity profile. We then introduce the hydrodynamic solution to the two-dimensional problem of alternating solid–water and air–water interfaces to determine the convective mass transfer of air's dissolution into water. This situation simulates spanwise microridges. The decoupled mass-transfer problem is solvable using an approximate integral method previously optimized by Reynolds et al. (1958). A mass-transfer correlation is derived as a function of the surface geometry (or gas area fraction), Reynolds number, and Schmidt number. Longevity, or time-dependent hydrophobicity, can be estimated from the resulting mass-transfer correlation. As expected, turbulence greatly enhances the rate of convective mass transfer, and thus superhydrophobicity is not maintained as long as it would under corresponding laminar flow conditions.

Monday, November 25, 2013 3:35PM - 5:45PM –

Session L21 Turbulence: Simulations V 316 - Qiqi Wang, Massachusetts Institute of Technology

3:35PM L21.00001 Least Squares Shadowing Sensitivity Analysis of Chaotic and Turbulent Fluid Flows[1] , PATRICK BLONIGAN, QIQI WANG, STEVEN GOMEZ, Massachusetts Institute of Technology — Computational methods for sensitivity analysis are invaluable tools for fluid dynamics research and engineering design. These methods are used in many applications, including aerodynamic shape optimization and adaptive grid refinement. However, traditional sensitivity analysis methods break down when applied to long-time averaged quantities in chaotic fluid flow fields, such as those obtained using high-fidelity turbulence simulations. This break down is due to the "Butterfly Effect"; the high sensitivity of chaotic dynamical systems to the initial condition. A new sensitivity analysis method developed by the authors, Least Squares Shadowing (LSS), can compute useful and accurate gradients for quantities of interest in chaotic and turbulent fluid flows. LSS computes gradients using the "shadow trajectory," a phase space trajectory (or solution) for which perturbations to the flow field do not grow exponentially in time. This talk will outline Least Squares Shadowing and demonstrate it on several chaotic and turbulent fluid flows, including homogeneous isotropic turbulence, Rayleigh-Bénard convection and turbulent channel flow.

[1]We would like to acknowledge AFSOR Award F11B-T06-0007 under Dr. Fariba Fahroo, NASA Award NNH11ZEA001N under Dr. Harold Atkins, as well as financial support from ConocoPhillips, the NDSEG fellowship and the ANSYS Fellowship.

3:48PM L21.00002 Towards Scalable Parallel-in-Time Turbulent Flow Simulations , QIQI WANG, STEVEN GOMEZ, PATRICK BLONIGAN, MIT, ALASTAIR GREGORY, Cambridge University, ELIZABETH QIAN, MIT — We present a reformulation of unsteady turbulent flow simulations that exhibits chaotic dynamics. Examples include many DNS and LES. This reformulation uses the concept of least squares shadowing. The flow field is assumed to be ergodic, and only long time averaged statistical quantities are considered as quantities of interest. The initial condition is relaxed in the least squares shadowing formulation, and information is allowed to propagate both forward and backward in time. Simulations of chaotic dynamical systems with this reformulation can be proven to be well-conditioned time domain boundary value problems. We analyze how this reformulation can enable scalable parallel-in-time simulation of turbulent flows.

4:01PM L21.00003 Classification of dense currents over rough walls[1] , RAGHIB CHOWDHURY, KIRAN BHAGANAGAR, University of Texas, San Antonio — Direct numerical simulations and RANS models have been used as a tool to simulate density currents over rough-walls consisting of cylindrical and sinusoidal roughness geometries with different spacing (λ) for given height (k) of roughness elements. Scaling laws of front velocity and locations in terms of the spacing between the roughness elements for sinusoidal shaped and sinusoidal roughness element have been obtained. Flow structures for sinusoidal roughness cases revealed that the wake generated at the valley region for sinusoidal or space between the cylinders plays a role on turbulent mixing which leads to reduction in frontal velocity. An important conclusion of the present study is different scaling exist for the k- type and d-type roughness.

[1]Acknowledge the support of NSF OCE-1333033

4:14PM L21.00004 FDF in US3D , COLLIN OTIS, University of Pittsburgh, PIETRO FERRERO, GRAHAM CANDLER, University of Minnesota, PEYMAN GIVI, University of Pittsburgh — The scalar filtered mass density function (SFMDF) methodology is implemented into the computer code US3D. This is an unstructured Eulerian finite volume hydrodynamic solver and has proven very effective for simulation of compressible turbulent flows. The resulting SFMDF-US3D code is employed for large eddy simulation (LES) on unstructured meshes. Simulations are conducted of subsonic and supersonic flows under non-reacting and reacting conditions. The consistency and the accuracy of the simulated results are assessed along with appraisal of the overall performance of the methodology. The SFMDF-US3D is now capable of simulating high speed flows in complex configurations.

4:27PM L21.00005 Three-dimensional simulation of slip-streaming in vehicle aerodynamics , SAURAV MITRA, Convergent Science Inc. — Simulation of slip-streaming in vehicle aerodynamics is computationally challenging. To resolve turbulent wakes, and estimate drag between two co-linear vehicles with less number of computational cells requires advanced techniques. In this study, the variation of drag reduction and increase arising due to different inter-spacing between two Ahmed vehicles bodies (canonical vehicle geometry with 30° slant back angle) are presented. The computational fluid dynamics solver CONVERGE was used, for its automatic mesh refinement (AMR) capabilities. AMR is based on the second derivative of shear and normal components of velocity gradients and was used to resolve the flow around geometric features such as the frontal area, the slant back, etc. Steady-state density-based solver is used where each cell has its own pseudo time-step based on the local numerical stability criterion. The RNG k-ε turbulence model was used to model turbulence. The non-dimensional inter-spacing based on vehicle length, was varied from 0.1 to 2.0. The largest grid size used here was 0.04 m and the smallest was 0.005 m to resolve the turbulent wake which is characterized by a strong vortex system, longitudinal counter-rotating vortices arising from the slant back.

4:40PM L21.00006 Turbulent transport at rough surfaces , SRIKANTH TOPPALADODDI, JOHN WETTLAUFER, Yale University, University of Oxford, SAURO SUCCI, Istituto Applicazioni Calcolo, CNR Roma — We use the Lattice Boltzmann Method to study the effects of rough walls on transport properties at large Reynolds numbers in two dimensions. The roughness elements used have both uniform and non-uniform distributions and we compare our approach with previous studies that have investigated the effects of rough walls on flows in micro channels. The non-uniform roughness distributions have the same spectral properties as that of the underside of Arctic sea ice.

4:53PM L21.00007 Stochastic field modeling of cavitating flows in OpenFOAM[1] , MICHAEL RANFT, Institute for nuclear and energy technology, Karlsruhe Institute of Technology, Germany, ANDREAS G. CLASS, Areva nuclear professional school, Karlsruhe Institute of Technology, Germany — In [1] analysis is presented for a fluidic diode with low/high pressure drop in forward/reverse flow direction. Accurate description of cavitation is needed due to the dominant effect of vapor bubbles on sound speed. The stochastic field method developed in [1] represents the statistics of growing cavitation bubbles by a set of stochastic fields of vapor fraction which evolve according to the Rayleigh-Plesset equation and local instantaneous LES flow conditions. Cavitation may originate from nucleation sites in the core of turbulent vortices. In this work a RANS model is used instead of LES. Local turbulent pressure fluctuations are recovered based on kinetic energy k of turbulence and its Dissipation ε. In the Rayleigh-Plesset equation these fluctuations are represented by a Wiener process which is superimposed on the mean pressure. Usually a set of stochastic fields is introduced for each stochastic variable. Here two independent Wiener processes, both acting on the vapor-fraction stochastic fields, drive the evolution of vapor bubble growth, so that a single set of stochastic fields can be maintained. The proposed methodology is implemented in OpenFOAM and applied to verification cases including the fluidic diode.

[1] Phys. Fluids 25, 073302 (2013).

[1]Funded by ANPS.

5:06PM L21.00008 Simulating 3D turbulence with Smoothed Particle Hydrodynamics , XIANGYU HU, STEFAN ADAMI, NIKOLAUS ADAMS, Technical University of Munich — In 2002 Monaghan showed a Lagrangian averaged SPH (Smoothed Particle Hydrodynamics) turbulence model and simulated two-dimensional turbulence. Although achieving good results, this method was shown to be computationally very inefficient (Monaghan, 2002). In this work we present results of 3D turbulence simulated with our newly developed weakly compressible SPH method with modified transport-velocity formulation (Adami, et al., 2013). This fundamental modification was first proposed by Monaghan (Monaghan, 1989). Different from XSPH, we solve a modified momentum equation including a constant background pressure field that regularizes particle motion "physically" while strongly reducing artificial numerical dissipation. Numerical results show that the dissipation rate of the 3D Taylor-Green vortex agrees well with DNS results and compared to the standard Smagorinsky model the accuracy is improved (as shown in Figure 1.). To the best knowledge of the authors, this is the first time that a weakly-compress SPH method achieves better results on turbulent flow than the standard grid-based model.

5:19PM L21.00009 Entropic Lattice Boltzmann Methods for Fluid Mechanics[1] , SHYAM CHIKATAMARLA, FABIAN BOESCH, DAVID SICHAU, ILYA KARLIN, ETH Zurich — With its roots in statistical mechanics and kinetic theory, the lattice Boltzmann method (LBM) is a paradigm-changing innovation, offering for the first time an intrinsically parallel CFD algorithm. Over the past two decades, LBM has achieved numerous results in the field of CFD and is now in a position to challenge state-of-the art CFD techniques. Our major restyling of LBM resulted in an unconditionally stable entropic LBM which restored Second Law (Boltzmann H theorem) in the LBM kinetics and thus enabled affordable direct simulations of fluid turbulence [2]. We review here recent advances in ELBM as a practical, modeling-free tool for simulation of turbulent flows in complex geometries. We shall present recent simulations including turbulent channel flow, flow past a circular cylinder, knotted vortex tubes, and flow past a surface mounted cube. ELBM listed all admissible lattices supporting a discrete entropy function and has classified them in hierarchically increasing order of accuracy[3]. Applications of these higher-order lattices to simulations of turbulence and thermal flows shall also be presented.

[1] Chikatamarla et al, J. Fluid. Mech, 656 (2010); Physica. A, 392 (2013)
[2] Chikatamarla and Karlin, Phys. Rev. Lett., 010201 (2006); Phys. Rev. Lett, 19060

[1]This work was supported CSCS grant s437.

5:32PM L21.00010 Efficient error estimation criteria to capture vortical structures in octree meshes , CANSU OZHAN, DANIEL FUSTER, PATRICK DA COSTA, CNRS (UMR 7190), Universite Pierre et Marie Curie, Institut Jean le Rond d'Alembert, France — This paper aims at finding optimal adaptive mesh refinement strategies to capture vortical structures. Due to their efficiency, we focus on a-posteriori mesh refinement methods. In particular, we derive a Hessian error estimator for the h-refinement scheme and a residual-based error estimator for finite volume methods and octree grids. The methods are validated for a classical test for the solution of the advection-diffusion-reaction equation and tested against three different test cases where vortical structures are present. In particular we test the temporal evolution of the Lamb-Oseen vortex, the linear growth-rate of small perturbations in a shear viscous layer and the energy evolution in the isotropic turbulence case. The performance of the proposed estimators and the choice of the optimal quantity of interest is discussed for different test cases.

Monday, November 25, 2013 3:35PM - 5:58PM –

Session L22 Turbulence: Mixing III 317 - Arne Johansson, KTH Royal Institute of Technology

3:35PM L22.00001 Large Eddy Simulation of Mixing within a Hypervelocity Scramjet Combustor , DAVID PETTY[1], VINCENT WHEATLEY[2], University of Queensland, CARLOS PANTANO[3], University of Illinois at Urbana-Champaign, MICHAEL SMART[4], University of Queensland — The turbulent mixing of parallel hypervelocity (U = 3230 m/sec, M = 3.86) air-streams with a sonic stream of gaseous hydrogen is simulated using large eddy simulation. The resultant mixing layers are characterized by a convective Mach number of 1.20. This configuration represents parallel slot injection of hydrogen via an intrusive centerbody within a constant area rectangular combustor. A hybrid shock-capturing/zero numerical dissipation (WENO/TCD) switch method designed for simulations of compressible turbulent flows was utilized. Sub-grid scale turbulence was modeled using the stretched vortex model. Visualizations of the three dimensional turbulent structures generated behind the centerbody will be presented. It has been observed that a span-wise instability of the wake behind the centerbody is initially dominant. Further downstream, the shear-layers coalesce into a mixing wake and develop the expected large-scale coherent span-wise vortices.

[1]Ph.D. Candidate, School of Mechanical and Mining Engineering, Centre for Hypersonics
[2]Senior Lecturer, School of Mechanical and Mining Engineering, Centre for Hypersonics
[3]Associate Professor of Mechanical Engineering
[4]Professor, School of Mechanical and Mining Engineering, Centre for Hypersonics

3:48PM L22.00002 On the effect of fractal generated turbulence on the heat transfer of circular impinging jets , TOMMASO ASTARITA, GIOACCHINO CAFIERO, STEFANO DISCETTI, Università degli Studi di Napoli Federico II — The intense local heat transfer achieved by circular impinging jets is exploited in countless industrial applications (cooling of turbine blades, paper drying, tempering of glass and metals, etc). The heat transfer rate depends mainly on the Reynolds number, the nozzle-to-plate distance and the upstream turbulence. It is possible to enhance the heat transfer by exciting/altering the large scale structures embedded within the jet. In this work turbulent energy is injected by using a fractal grid at the nozzle exit. Fractal grids can generate more intense turbulence with respect to regular grids with the same blockage ratio by enhancing the jet turbulence over different scales. Consequently, they are expected to improve the convective heat transfer. The results outline that a significant improvement is achieved (for small nozzle-to-plate distances up to 100% at the stagnation point and more than 10% on the integral heat transfer over a circular area of 3 nozzle diameters) under the same power input.

4:01PM L22.00003 Quantification of Mixing of a Sonic Jet in Supersonic Crossflow due to Thick Turbulent Boundary Layer Interaction , TOBIAS ROSSMANN, Lafayette College, ADAM PIZZAIA, Rutgers University — The upstream injection surface boundary layer is shown to have a significant effect on the mixing characteristics of a sonic jet in supersonic cross flow. A circular, high-pressure, sonic jet is injected into a M=3.5 supersonic crossflow through different boundary layer thickness (δ/D = 7.5 and 1), with variable injection angles (-20 to +20 degrees), and variable momentum ratios (J = 2, 5, and 10). Planar Laser Mie Scattering of condensed ethanol droplets is used to quantitatively image the injected fluid concentration in both the side and end views. Jet fluid concentrations PDFs are constructed to better understand the mixing dynamics. These PDFs are integrated to create mixed fluid fraction profiles that are then reduced to mixing efficiency. Mixing efficiency values are computed from different two-dimensional planes to determine if centerline mixing efficiencies are characteristic of the entire three-dimensional flow. Through these analyses, it is seen that thick boundary layers tend to marginally alter jet penetration and spread, but significantly worsen jet mixing capabilities, regardless of momentum ratio or injection angle.

4:14PM L22.00004 Turbulent Mixing of an Angled Jet in Various Mainstream Conditions , KEVIN RYAN, FILIPPO COLETTI, CHRISTOPHER ELKINS, JOHN EATON, Stanford University — The angled jet in crossflow has been studied in detail with specific emphasis on the turbulent mixing of the jet fluid with the mainstream flow. The interaction of the upstream boundary layer with the jet shear layer results in complex vortex patterns that cause large mean distortion of the jet and rapid turbulent mixing. Most previous studies have been conducted in flat plate flows with little attention paid to the characteristics of the boundary layer. The present study examines the effect of mainstream geometric changes on the jet trajectory, counter-rotating vortex pair strength, and turbulent mixing. Seven cases were examined including flat plate boundary layers with three different thicknesses, adverse and favorable pressure gradient cases, and flows with concave and convex streamwise curvature. Full field, 3D mean velocity and scalar concentration fields were measured using magnetic resonance imaging (MRI) techniques in a water flow. The distortion of the streamtube initiated at the hole exit was examined for each of the seven cases. The degree of mixing was quantified by measuring the amount of mainstream fluid entrained into the jet as well as the turbulent diffusivity as a function of streamwise position.

4:27PM L22.00005 Interaction of Inflow Jet and Flow Recirculation in Large Mixing Tanks , MARKUS VAAS, Institute for Hydromechanics, Karlsruhe Institute of Technology, EMMA THOMPSON, University of Guelph, JOCHEN KRIEGSEIS, Institute of Fluid Mechanics, Karlsruhe Institute of Technology — Jet inflow into a confined box-shaped domain (tank) can induce a slow, stable recirculation flow. Once developed, these large scale structures compete with the spatial distribution of the inflow jet. The objective of the present work is to understand the underlying mechanisms which lead to this interaction/competition of the respective flow patterns. Particular emphasis is placed on the jet's self-similarity and its axial velocity decay, since both are of prime importance for the resulting entrainment along the jet borders and the overall mixing efficiency in the tank. Particle image velocimetry (PIV) measurements have been performed in a tank with turbulent inflow jets of varying jet Reynolds numbers. Coherent large-scale structures superimposed to the aforementioned mean flow fields were identified by means of proper orthogonal decomposition (POD). As such, both the meandering character of the jet core as well as fluctuating patterns of the recirculation zone have been identified. The impact of the confinement is demonstrated by a direct comparison of the POD results with the most salient free-jet patterns. Based on these insights, implications for entrainment and mixing stemming from the interaction of the jet and the recirculating flow are discussed.

4:40PM L22.00006 Dynamics of spinodal decomposition in turbulent flows , FEDERICO TOSCHI, Eindhoven University of Technology, The Netherlands, ROBERTO BENZI, University of Rome "Tor Vergata", Rome, Italy, HERMAN CLERCX, Eindhoven University of Technology, The Netherlands, DAVID NELSON, Harvard University, Cambridge, USA, PRASAD PERLEKAR, TIFR, Hyderabad, India — When a binary mixture is cooled below its critical temperature it undergoes a phase transition and the mixture separates into its individual components: this phenomenon is widely known as spinodal decomposition. The dynamics proceeds through different regimes all characterized by a coarsening of the domain size. We investigate numerically the dynamics of such a system when the mixture of immiscible fluids is stirred at the large scale and thus turbulent. Under turbulent conditions we find that the coarsening of the domains is arrested and a similarity with the physics of dilute turbulent emulsions is possible. In particular we show that the typical domain size can be estimated by means of the Kolmogorov-Hinze argument for the stability of droplets in turbulence.

4:53PM L22.00007 Free shearless multi-material turbulent mixing in the presence and absence of gravity[1] , POOYA MOVAHED, ERIC JOHNSEN, University of Michigan, Ann Arbor — Using a novel set-up, we perform direct numerical simulations of free shearless turbulent multi-material mixing starting from an unperturbed material interface between two fluids in an isotropic turbulent velocity field. The energy dissipation rate is matched in each fluid, such that anisotropy in the initial set-up solely comes from the density gradient. At large scales, the mixing region grows self-similarly after an initial transient period; a one-dimensional turbulence-diffusion model in conjunction with Prandtl's mixing length theory is applied to describe the growth of the mixing region. The observed growth exponent tends to 2/7, as expected for Batchelor turbulence based on energy budget arguments for large Reynolds numbers. At small scales, flow isotropy and intermittency are measured. Results suggest that a large density ratio between the two fluids is required to make the velocity field anisotropic at the Taylor microscope, while the flow remains isotropic at the Kolmogorov microscale. Results with gravity in a similar set-up that is Rayleigh-Taylor unstable will also be presented. This novel set-up allows us to investigate the role of gravity and density gradient on flow statistics separately, as opposed to traditional Rayleigh-Taylor studies in which these effects are coupled.

[1]This research was supported in part by the DOE NNSA under the Predictive Science Academic Alliance Program by grant DEFC52- 08NA28616.

5:06PM L22.00008 Turbulent Buoyant Flows: A Structural Approach , PHARES CARROLL, GUILLAUME BLANQUART, California Institute of Technology — In a wide variety of natural phenomena, buoyancy plays a significant role in the evolution of turbulent physics. However, there is no consensus in the literature in regards to the structural implications of the interplay between turbulence and buoyancy. In this study, two incompressible, miscible fluids with different densities are considered in the presence of gravity. In the implemented configuration, it is possible to vary independently the Reynolds and Richardson numbers, allowing for the methodical study of the interactions between buoyancy and turbulence. The simulation results are used to analyze the structural changes induced by buoyancy on the evolving turbulent field. Specifically, attention is paid to the spectral distribution of kinetic energy via calculation and analysis of spectra (energy production, dissipation, and transfer). This is done to determine where gravity (buoyancy) deposits its energy from a spectral perspective. Also, the evolution of each constituent term in the energy equation is examined and compared to results obtained from isotropic turbulent cases. Further, analysis is conducted on the alignment of the vorticity field with the direction of principle strains to identify how the turbulent structure is modified in the presence of such a body force.

5:19PM L22.00009 Turbulent fountains as a model for mixing at a density interface , NIGEL KAYE, WILLIAM MARTIN III, Clemson University — Numerical results are presented for the change in flow rate in a round turbulent fountain as it rises and falls. The theoretical models of Bloomfield and Kerr (2000) are used to examine the entrainment across a density interface due to the impingement of an axisymmetric turbulent shear flow such as a plume or jet. The numerical results indicate that there are two mixing regimes, a low Froude number (weak fountain) regime in which the entrainment coefficient scales approximately on $\alpha \sim Fr^2$ and a high Froude number (forced fountain) regime in which the entrainment coefficient scales like $\alpha \sim Fr$. This is in contrast to the experimental parameterization of Kumagai (1984) which had the entrainment coefficient as $\alpha \sim Fr^3$ for low Froude numbers and α approaching a constant in the limit of high Fr. The numerical results are compared to a broad range of experimental measurements in the literature. The sensitivity of the calculated entrainment coefficients to the internal body force model and entrainment assumption is also examined. The results are discussed in relation to models for mixing efficiency in stratified flows.

5:32PM L22.00010 Experimental variable-density mixing statistics , SERGIY GERASHCHENKO, KATHERINE PRESTRIDGE, Los Alamos National Laboratory — Velocity and density statistics are studied experimentally for variable density mixing of a heavy fluid jet into air coflow at two Atwood numbers. The effect of buoyancy is found to be important for most turbulent quantities measured. The high At jet with larger Reynolds number shows reduced lateral spreading compared to the low At jet of smaller Reynolds number. Some universal features of variable density mixing are elucidated from PDFs of density and density gradients. The low Atwood number PDFs show fast and uniform mixing. High Atwood number PDFs of density have skewness toward the larger densities, indicating reduced rate of mixing of pure heavy fluid due to its inertia. This skewness is related to strong local compression events that can lead to enhanced molecular mixing. Turbulent kinetic energy decreases with distance from the jet for low Atwood number but increases for high Atwood number due to flow acceleration and generation of extra shear and turbulence. This is clearly a buoyancy-mediated effect. Statistical characteristics of mixing such as Favre-averaged Reynolds stress and its anisotropy, turbulent mass flux velocity, density-specific volume correlation, density power spectra are also examined in the near and far field from the jet.

5:45PM L22.00011 Experiments on the fragmentation of a buoyant liquid volume in another liquid , MAYLIS LANDEAU, Geological Fluid Dynamics, IPGP, France, RENAUD DEGUEN, IMFT, France, PETER OLSON, Johns Hopkins University, USA — Buoyancy-driven fragmentation of one liquid in another immiscible liquid was a common process during the formation of the terrestrial planets. Another example of this phenomenon is the sudden release of petroleum into the ocean during the Deepwater Horizon disaster. In this study, we present experiments on the instability and fragmentation of volumes of heavier liquid released into lighter immiscible liquids. We characterize the different fragmentation regimes in parameter space. We find that, at low and intermediate Weber numbers (measuring the importance of inertia versus surface tension forces), the fragmentation regime mainly results from a competition between the growth of Rayleigh-Taylor instabilities and the roll-up of a vortex ring. At high Weber numbers, a turbulent fragmentation regime is found, and the large-scale flow behaves as a turbulent vortex ring or a turbulent thermal. An integral model based on the entrainment assumption, and adapted to buoyant vortex rings with initial momentum, is consistent with our experimental data. This indicates that the concept of turbulent entrainment is valid for non-dispersed immiscible fluids at large Weber and Reynolds numbers.

Monday, November 25, 2013 3:35PM - 5:45PM –

Session L23 Turbulence: Theory V - Measurements 318 - Greg Voth, Wesleyan University

3:35PM L23.00001 Extracting Turbulent Spectral Transfer from Under-Resolved Velocity Fields , NICHOLAS OUELLETTE, Yale University, RUI NI, GREG VOTH, Wesleyan University — The strong nonlinearities in turbulent flows drive the transfer of energy and other quantities among different scales of motion. In 3D turbulence, this transfer organizes into the classic Richardson-Kolmogorov cascade of energy to small scales; in 2D turbulence, it leads to an inverse cascade of energy to large scales and a forward cascade of enstrophy to small scales. Directly measuring this spectral transfer is difficult, particularly in experiments. Recent developments of filtering techniques allow spectral fluxes to be measured locally, but have been assumed to require finely resolved velocity fields that are typically not available in 3D experiments. Here we show, using experimental data in 2D and DNS data in 3D, that poorly resolved velocity fields can still be used to extract information about spectral transfer processes. Our results also have implications for locality in the cascades.

3:48PM L23.00002 Decaying turbulence in the presence of a shearless uniform kinetic energy gradient[1] , ADRIEN THORMANN, CHARLES MENEVEAU, Johns Hopkins University — The study of decaying isotropic turbulent flow is an important point of reference for turbulence theories and numerical simulations. For the past several decades, most experimental results have focussed on possible power-law decays and found exponents between -1 and -1.4, approximately. Another class of experiments have been shear less mixing layers in which there are two regions with different kinetic energy levels that slowly diffuse into each other downstream. In this study we consider flow without shear-production of turbulence with a cross-stream uniform spatial gradient of kinetic energy k(z) = C z. Such gradient is generated with the use of an active grid and screens mounted upstream of the wind-tunnel's test section iteratively designed to produce a linear gradient of kinetic energy without mean shear. In such a flow, deviations from constant lateral flux of kinetic energy are due only to spatial variations in turbulent diffusivity of k (turbophoresis). Data are acquired using X-wire thermal anemometry at different spanwise and downstream locations. Tests of homogeneity, as well as spectral characteristics of the flow, decay and diffusion rates of the kinetic energy will be presented.

[1]This research is supported by NSF-CBET-1033942.

4:01PM L23.00003 Multi-scale grid generated turbulence in an internal flow application , PIYUSH RANADE, SCOTT MORRIS, University of Notre Dame — Turbulence generation using multi-scale, or fractal grids, is a method of creating high turbulence intensity flows passively by utilizing the intrinsic scales associated with the grid. This has become the topic of research in many external flow applications. In turbomachinery, the flow at the exit of the combustor and into the first nozzle stage is highly turbulent. In order to create high turbulence intensities in a lab setting passively, multi-scale turbulence generation grids are proposed. The presence of multiple length scales in the grid geometry innately gives rise to turbulent motions of a wide spectrum being shed immediately downstream of the grid, leading to high turbulence intensity flow. The biggest challenge with using such a grid in an internal flow, however, is to achieve spatial uniformity. In this research, three grid geometries commonly found in literature were tested in an experimental set-up consisting of flow between two flat plates. In addition, several other fractal grid geometries were created and tested in an attempt to maximize turbulence intensity while maintaining spatial homogeneity. This research hopes to begin giving insight into the development of turbulence downstream of a multi-scale grid in an internal flow setting.

4:14PM L23.00004 Alignment of two-point statistics with respect to mean deformation field in anisotropic turbulent flows , KHANDAKAR MORSHED, LAKSHMI DASI, Colorado State University — We study the variations in two-point correlation functions and second-order structure functions in the strongly anisotropic turbulent flow past a backward facing step. Time-resolved particle image velocimetry measurements were performed in a stationary turbulent flow past a backward facing step at Reynolds numbers 13,600, 9,000, and 5,500 based on the maximum velocity and step size. Measurements revealed a strongly anisotropic large-scale flow with an intense turbulent free-shear layer downstream of the step. Comparison among local two-point correlation functions and second-order structure functions at varying locations within the measurement domain reveals a mechanistic relationship between the magnitude of mean flow deformation field and the spatial organization of the two-point statistics in 360 degrees. It is shown that the local spatial variation in rms velocity significantly induces local anisotropy at arbitrarily small length scales.

4:27PM L23.00005 Reconstruction of Coherent Structures from Time Shifted Data , ANDREW NEWMAN, LUCIANO CASTILLO, Texas Tech University — The possibility of extracting coherent structure information from experimental data which has a time shift is considered. The presented techniques are especially applicable to particle image velocimetry (PIV) data collected from large experiments where all PIV planes cannot be collected simultaneously. The method relies on the Proper Orthogonal Decomposition and considers ways to construct POD modes for two adjacent data domains (but with a time shift between them) out of POD modes computed from the individual domains where all data is collected simultaneously. It is first shown that such composite modes exist. Further, it is shown through example how this procedure is done. Lastly, it is shown that the technique is applicable to experimental data though the problem becomes increasing complex.

4:40PM L23.00006 Scale-by-scale energy transfer in the production region of a fractal grid , RAFAEL GOMES 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 time-resolved Particle Image Velocimetry in a water tunnel. Scale-by-scale energy transfer is computed using the transport equation of the second order structure function for inhomogeneous flows. The balance of each of the equation terms is presented. In some particular locations in the production region, the map of the radial divergence of the energy flux shows an upward energy transfer in the direction of the mean flow and downwards in the perpendicular direction. In these locations, the energy spectra already exhibits a well-defined -5/3 power law over more than one decade; and the third order structure function of the velocity component parallel to the mean flow, evaluated in that direction at the centerline, has a positive sign throughout the range of scales dominated by the -5/3 scaling. Any velocity derivative checks of small-scale isotropy available to us indicate that the small-scales are indeed isotropic in these locations.

4:53PM L23.00007 Fluctuations in the energy input determine Kolmogorov constants in turbulence , GREGORY BEWLEY, Max Planck Institute for Dynamics and Self-Organization, FLORENT LACHAUSSÉE, École Normale Supérieure, JOHANNES KASSEL, University of Göttingen, GREG VOTH, Wesleyan University, EBERHARD BODENSCHATZ, Max Planck Institute for Dynamics and Self-Organization — Attention to turbulence is often focused, for good reason, on flows that either maintain a steady state or decay freely. But these conditions are not typical in natural or industrial flows. We ask what effect deviations from these conditions have on the turbulence itself. To answer the question, we employ a new active grid with many independently controllable degrees of freedom to generate turbulence in a wind tunnel. We find the following: The anisotropy in the flow can be set to various states, including an isotropic one, by adjusting the correlations between motions on the grid. Some part of the fluctuations in the flow can be attributed to the instantaneous configuration of the grid, in the sense that it is reproduced when the grid returns to the same configuration. The value of the Kolmogorov constants for the structure functions of different order can be adjusted by changing over time the degree to which the active grid agitates the flow. We interpret these variations in agitation as variations of the energy input rate. We then find that the Kolmogorov constants, in particular those of order higher than two, can be made to have universal values when the variation of the energy input rate is accounted for by a model based on the refined similarity theory.

5:06PM L23.00008 Inertial range ESS scaling deteriorates with increasing Reynolds number , EBERHARD BODENSCHATZ, MICHAEL SINHUBER, GREGORY BEWLEY, Max Planck Institute for Dynamics and Self-Organization, MARGIT VALLIKIVI, MARCUS HULTMARK, ALEXANDER SMITS, Princeton University — We examined the scaling of velocity structure functions in turbulence generated by a classical biplanar grid of crossed bars in the Variable Density Turbulence Tunnel in Göttingen. The flow had neither a mean shear nor strong anisotropy. Despite this, the structure functions did not exhibit power-law scaling unless Extended Self-Similarity (ESS) was employed. The ESS exponents were remarkably stable at Taylor Reynolds numbers between 100 and 1600. That is, at higher Reynolds numbers than in any other comparable flow. However, the extent to which ESS applied at small scales deteriorated as the Reynolds number increased. The experiments were performed in pressurized sulfur hexafluoride gas at pressures between 1 and 15 bar. The data were acquired with both classical hot wires, and with the NSTAP anemometers developed at Princeton.

5:19PM L23.00009 Detrended analysis of Reynolds stress in a decaying turbulent flow in a wind tunnel with active grids , ZHIMING LU, IMTIAZ AHMAD, YONGXIANG HUANG, Shanghai Institute of Applied mathematics and Mechanics, Shanghai University — Multi-scale properties of Reynolds stress in decaying turbulence at a wind tunnel with high Reynolds number are investigated. Two filtering technique i.e., zeroth-order and first-order detrending method are applied to the two velocity components, where the local mean value (resp. local linear trend) is removed in the former (latter) technique. Some basic statistics for thirty measurements show that the variation is very large at first two locations and relatively small at last two locations. Moderately good power law is found for the mean value of local Reynolds stress at last three measurement locations with scaling exponents approximately being 1.0 and a dual power law exists for the mean value of standard deviation of local Reynolds stress at all four measurement locations with scaling exponents being 0.53 and 0.58 for zeroth- and first-order filtering respectively.

5:32PM L23.00010 Anomalous scaling of passive scalar fluctuations in a spatially developing turbulent mixing layer , ANTONIO ATTILI, FABRIZIO BISETTI, King Abdullah University of Science and Technology — A detailed statistical analysis of fluctuations of a passive scalar in the fully developed region of a spatially developing turbulent mixing layer from a direct numerical simulation is presented. Passive scalar spectra show inertial ranges characterized by scaling exponents $-4/3$ and $-3/2$ in the streamwise and spanwise directions, in agreement with recent theoretical analysis of passive scalar scaling in shear flows [Celani *et al.* J. Fluid Mech. 523, 99 (2005)]. Scaling exponents of high-order structure functions in the streamwise direction show saturation of intermittency with an asymptotic exponent $\zeta_\infty = 0.4$ at large orders. Saturation of intermittency is confirmed by the self-similarity of the tails of the probability density functions of the scalar increments at different scales r with the scaling factor $r^{-\zeta_\infty}$ and by the analysis of the cumulative probability of large fluctuations. Conversely, intermittency saturation is not observed for the spanwise increments, and the relative scaling exponents agree with recent results for homogeneous isotropic turbulence with mean scalar gradient. Probability density functions of the scalar increments in the three directions are compared to assess anisotropy.

Monday, November 25, 2013 3:35PM - 6:11PM –

Session L24 Aerodynamics IV 319 - Sunil James, Honeywell Aerospace

3:35PM L24.00001 On The Flow Physics of Dynamic Stall Inception , DUSTIN COLEMAN, FLINT THOMAS, KYLE HEINTZ, MICHAEL WICKS, THOMAS CORKE, University of Notre Dame — Despite being the focus of many previous investigations, dynamic stall inception is still not fully understood. In this study the flow physics regarding the initiation, growth, and convection of the dynamic stall vortical structure produced during unsteady pitching of a NACA 0015 airfoil are investigated using time-resolved particle imaging velocimetry (TR-PIV) and surface pressure measurements as the primary flow diagnostics. Experiments are conducted at freestream Mach and Reynolds numbers of $M_\infty = 0.1$ and $\mathrm{Re}_c = 2.75\mathrm{e}{+05}$, respectively, and over a reduced frequency range of $0.05-0.1$. The experimental measurements are input to a locally implemented control volume formulation in order to characterize the convection and wall-normal diffusion of spanwise vorticity near the leading edge. In this manner, the near-wall flow physics surrounding dynamic stall vortex (DSV) inception is characterized. Likewise, the evolution of the resultant DSV is characterized in terms of its growth rate, terminal strength, and wall detachment process.

3:48PM L24.00002 Reduced-order vortex modeling of unsteady non-linear aerodynamics[1] , JEFF ELDREDGE, DARWIN DARAKANANDA, Mechanical and Aerospace Engineering, University of California, Los Angeles, MAZIAR HEMATI, Mechanical and Aerospace Engineering, Princeton University — Non-linear fluid dynamic phenomena are inherent both to flapping wings and to fixed wings during rapid maneuvers. These phenomena, manifested in the interactions of shed vortex structures, are central to the generation of forces and moments. In previous work, we have presented the development and optimization of a low-degree-of-freedom model that captures such phenomena in the motions of point vortices of time-varying strength. Here, we present several extensions of this model toward more complex physics. The model construction is informed from a combination of results from experiments, high-fidelity Navier-Stokes computations, and inviscid vortex sheet simulations. A window-stitching technique is used to develop optimized point vortex models for longer-duration maneuvers. Self-sustained vortex shedding from a wing at large angle of attack is captured with point vortices – one per shed vortical structure – using a simple criterion based on the dynamics of the re-attachment point. Finally, the ongoing extension of the model to finite aspect ratio wings is presented.

[1]Supported by AFOSR award number FA9550-11-1-0098

4:01PM L24.00003 Physical model of kitesurfing , PAWEL ZIMOCH, ADAM PAXSON, EDWARD OBROPTA, TOM PELEG, SAM PARKER, A.E. HOSOI, Massachusetts Institute of Technology — Kitesurfing is a popular water sport, similar to windsurfing, utilizing a surfboard-like platform pulled by a large kite operated by the surfer. While the kite generates thrust that propels the surfer across the water, much like a traditional sail, it is also capable of generating vertical forces on the surfer, reducing the hydrodynamic lift generated by the surfboard required to support the surfer's weight. This in turn reduces drag acting on the surfboard, making sailing possible in winds lower than required by other sailing sports. We describe aerodynamic and hydrodynamic models for the forces acting on the kite and the surfboard, and couple them while considering the kite's position in space and the requirement for the kite to support its own weight. We then use these models to quantitatively characterize the significance of the vertical force component generated by the kite on sailing performance (the magnitude of achievable steady-state velocities and the range of headings, relative to the true wind direction, in which sailing is possible), and the degradation in sailing performance with decreasing wind speeds. Finally, we identify the areas of kite and surfboard design whose development could have the greatest impact on improving sailing performance in low wind conditions.

4:14PM L24.00004 Compressible flow in fluidic oscillators[1] , EMILIO GRAFF, DAMIAN HIRSCH, MORY GHARIB, Caltech — We present qualitative observations on the internal flow characteristics of fluidic oscillator geometries commonly referred to as sweeping jets in active flow control applications. We also discuss the effect of the geometry on the output jet in conditions from startup to supersonic exit velocity.

[1]Supported by the Boeing Company

4:27PM L24.00005 Lift generation on a flat plate with unsteady motions , XI XIA, KAMRAN MOHSENI, University of Florida — The leading edge vortex (LEV) on an airfoil or wing has been considered to be one of the most important sources of lift enhancement according to several previous experimental and theoretical studies. In this work, the unsteady 2D potential flow theory is employed to model the flow field of a flat plate wing undergoing unsteady motions. A multi-vortices model is developed to model both the leading edge and trailing edge vortices (TEVs), which offers improved accuracy compared with using only single vortex at each separation location. The lift prediction is obtained by integrating the unsteady Blasius equation. It is found that the motion of vortices contributes significantly to the overall aerodynamic force on the flat plate. The results of the simulation are then compared with classical numerical, theoretical and experimental data for canonical unsteady flat plat problems. Good agreement with these data is observed. Moreover, these results suggests that the leading edge vortex shedding for small angles of attack should be modeled differently than that for large angles of attack. Finally, the results of vortex motion vs. lift indicate that the lift enhancement during the LEV "stabilization" above the wing is a combined effect of both the LEV and TEV motion.

4:40PM L24.00006 Flow Visualization around a Simplified Two-Wheel Landing Gear , ALIS EKMEKCI, GRAHAM FELTHAM, University of Toronto — The flow topology around a simplified two-wheel landing gear model is investigated experimentally by employing the hydrogen bubble flow visualization technique in a recirculating water channel. The landing gear test model consists of two identical wheels, an axle, a main strut and a support strut. The flow Reynolds number based on wheel diameter is 31,500 and wheels with varying geometric details are considered. Flow structures have been identified through analysis of long-time video recordings and linked to the model geometry. In the flow region above the wheels (wing side), the flow in the inter-wheel region either separates prematurely from the inner surfaces of the wheels and forms slant vortices in the near-wake, or remains attached till the aft wheel perimeter. Inclusion of interior wheel wells are found to result in a jet-like ejection as a result of the interaction with the axle and main strut. In the flow region below the wheels (ground side) the near wake contains periodically forming, complex, large-scale structures.

4:53PM L24.00007 Tip vortex characteristics of rotor in hover , SWATHI M. MULA, CHRISTOPHER G. CAMERON, CHARLES E. TINNEY, JAYANT SIROHI, The University of Texas at Austin — Vortices emanating from the tip of the rotor blades comprise four distinct regions of flow: laminar, transitional, turbulent, and irrotational flows. To investigate the structural instabilities associated with various flow regions within the vortex, the current investigation employs the proper orthogonal decomposition (POD) technique. This technique is applied to blade tip vortices emanated from a reduced-scale, 1.0 m diameter, single-bladed rotor in hover. The rotor is operated at 1500 RPM which corresponds to a $Re_{tip} = 218{,}000$ and $M_{tip} = 0.23$; and at a collective pitch angle of 7.3°. Measurements are undertaken using a two-component PIV system, at various vortex ages. An effort is also made to ensure that there is sufficient resolution within the tip vortex region, to enable the study of local instabilities associated with various flow regions within the vortex.

5:06PM L24.00008 Perching Dynamics and Development of a Simple Model , MICHAEL PUOPOLO, JAMEY JACOB, Oklahoma State University, RYAN REYNOLDS, Sandia National Laboratory — Aerodynamicists with a vision for bird-like aircraft have been forced to develop new ways of modeling extremely agile flight systems, and in recent years there has been a growing variety of creative approaches that incorporate computer methods, empirical data, and unsteady flow theory. However, there remains a lack of simple and easily transferable models that can be used to predict and control motion of a fixed-wing, perching aircraft in the low Reynolds number flow regime. The authors have developed a simple dynamic model for a perching vehicle with a common fixed wing configuration that uses only input of the system design parameters, in addition to other relevant widely available information, and does not rely on wind tunnel measurements, CFD analysis or other rigorous forms of system identification. The resulting model is presented with a comparison of model simulations to flight data from a perching UAV.

5:19PM L24.00009 Development of a MEMS shear stress sensor for use in wind tunnel applications[1] , CASEY BARNARD, JESSICA MELOY, MARK SHEPLAK, University of Florida, INTERDISCIPLINARY MICROSYSTEMS GROUP TEAM — The measurement of mean and fluctuating wall shear-stress in laminar, transitional, and turbulent boundary layers and channel flows has applications both in industry and the scientific community. Currently there is no method for time resolved, direct measurement of wall shear stress at the spatial and temporal scales of turbulent flow structures inside model testing facilities. To address this need, a silicon micromachined differential capacitance shear stress sensor system has been developed. Mean measurements are enabled by custom synchronous modulation/demodulation circuitry, which allows for measurement of both magnitude and phase of incident wall shear stress. Sizes of the largest device features are on the order of relevant viscous length scales, to minimize flow disturbance and provide a hydraulically smooth sensing surface. Static calibration is performed in a flow cell setup, and an acoustic plane wave tube is used for dynamic response data. Normalized sensitivity of 1.34 mV/V/Pa has been observed over a bandwidth of 4.8 kHz, with a minimum detectable signal of 6.5 mPa. Initial results show qualitative agreement with contemporary measurement techniques. The design, fabrication, support electronics, characterization, and preliminary experimental performance of this sensor will be presented.

[1]The support of NASA SFW-NRA NNX11AI30A, AFOSR grant #FA 9550-12-1-0469, and Sandia Campus Executive Fellowship are gratefully acknowledged.

5:32PM L24.00010 Flow structure on a rotating wing undergoing deceleration to rest , DANIEL TUDBALL SMITH, Monash University, DONALD ROCKWELL, Lehigh University, JOHN SHERIDAN, Monash University — Inspired by the behavior of small biological flyers and micro aerial Vehicles, this study experimentally addresses the flow structure on a low aspect ratio rotating wing at low Reynolds number. The study focuses on a wing decelerating to rest after rotating at constant velocity. The wing was set to a constant 45° angle of attack and, during the initial phase of the motion, accelerated to a constant velocity at its radius of gyration, which resulted in a Reynolds number of 1400 based on the chord length. Stereoscopic PIV was used to construct phase-averaged three-dimensional (volumetric) velocity fields that develop and relax throughout the deceleration and cessation of the wing motion. During gradual deceleration, the flow structure is maintained when normalised by the instantaneous velocity; the distinguishing feature is shedding of a trailing edge vortex that develops due to the deceleration. At higher deceleration rates to rest, the flow structure quickly degrades. Induced flow in the upstream direction along the surface of the wing causes detachment of the previously stable leading edge vortex; simultaneously, a trailing-edge vortex and the reoriented tip vortex form a co-rotating vortex pair, drawing flow downward away from the wing.

5:45PM L24.00011 Flow Structure on a Wing Due to Unsteady Pitch-Up and Rotation Maneuvers , MATTHEW BROSS, TURGUT YILMAZ, DONALD ROCKWELL, Lehigh University — The flow structure along a rectangular (low aspect ratio) wing undergoing pure pitch-up, pitch-up with rotation, and pure rotation is characterized as a function of dimensionless convective time τ during each maneuver. Quantitative imaging via angular displacement stereo particle image velocimetry was used to determine the three-dimensional velocity field, thereby allowing analysis of the effects of different wing kinematics via representations of Q-criterion, vorticity flux, and velocity and vorticity contours. Despite the difference in wing kinematics, interactions between leading-edge and tip vortices persist across all values of τ. The three-dimensional flow structure involves a symmetric pattern along the wing during pure pitch-up and transforms to a conical leading-edge vortex in conjunction with a tip vortex that extends into the wake for both pitch-up with rotation and pure rotation. This observation suggests that rotational motion has a greater influence than pitching motion in establishing the form and scale of the leading-edge vortex. Finally, sectional images of the flow structure arising from combined pitch-up and rotation were acquired at three different pitch rates relative to a given rate of pure rotation at fixed angle of attack.

5:58PM L24.00012 The Speed Mach 20 is Quite Impossible in Atmosphere! How to Calculate the Speed Limit When Accelerating an Object in Atmosphere , MWIZERWA PIERRE CELESTIN, None — This paper aims to respond to different mysteries which appear in news and blogs around the world about hypersonic flights, Colombia disaster, and hypersonic tests which are going on nowadays. People are really confused and wonder if, when we travel at hypersonic speed, the laws of physics change or if they remain intact. The laws of physics never change. It is we, humans, who have to respect those laws. In this paper, I will try to demonstrate and do all necessary calculations so people may find what is going on.

Monday, November 25, 2013 3:35PM - 5:45PM –

Session L25 Vortex Dynamics and Vortex Flows VIII 320 - Robert Krasny, University of Michigan

3:35PM L25.00001 Investigation of turbulent energy transport by applying POD-LSE complementary method[1] , OSAMU TERASHIMA, YASUHIKO SAKAI, KOUJI NAGATA, YASUMASA ITO, Nagoya University — Turbulent energy transport mechanism involving large-scale coherent vortex structures in the self-similarity region of a plane turbulent jet is experimentally investigated. First, a simultaneous multipoint measurement of two velocity components and pressure is performed by using several combined probes consisting of a pressure probe and an X-type hot-wire probe. Then, proper orthogonal decomposition (POD) is applied to both velocity and pressure fields to determine the coherent vortex structure in the jet. Further, a complementary technique of POD and linear stochastic estimation (LSE) is used to reconstruct the spatiotemporal velocity and pressure field of the dominant POD mode. As a result of reconstruction, the coherent structure with counter-rotating vortices staggering to the jet centerline is extracted and it contains approximately 42% of the total turbulent energy. Finally, the turbulent energy transport caused by the large scale coherent vortex structure is evaluated by using the reconstructed velocity and pressure fields. The results show that the production and pressure diffusion of the turbulent energy in the jet are mainly caused by this coherent vortex structure.

[1]Grants-in-Aid for scientific research (No. 23760155) and Takahashi Industrial and Economic Research Foundation

3:48PM L25.00002 Formation of Small-Scale Vortex Rings from Vortex Pairs Close to the Ground , DANIEL ASSELIN, CHARLES WILLIAMSON, Cornell University — In this research, we examine the effect of a solid boundary on the dynamics and instabilities of a pair of counter-rotating vortices. An isolated vortex pair is subject to a short-wave elliptic instability and a long-wave Crow (1970) instability. Near a wall, the boundary layer between the primary vortices and the wall can separate, leading to the generation of secondary vorticity. These secondary vortices can be subject to small-scale instabilities (Harris & Williamson, 2012) as they come under the influence of the primary vortices. In contrast, in the present study we are interested in the long-wave Crow instability interrupted by interaction with a wall. This can cause significant axial flow, resulting in a periodic concentration of fluid containing vorticity at the peaks of each wavy vortex tube and a corresponding evacuation of fluid containing vorticity from the troughs. It appears that this axial flow is driven at least in part by the formation of vortex ring-like structures in the secondary vortex as it is deformed by the primary vortex. Furthermore, additional small scale-vortex rings evolve from the secondary vorticity and from the concentrated vortical regions in the primary vorticity. In both cases, these rings cause vorticity to rebound away from the ground.

4:01PM L25.00003 Vortex wandering in grid turbulence[1] , STEFFEN PENTELOW, STAVROS TAVOULARIS, University of Ottawa — The tip vortex of a square-tipped NACA 0012 wing at an angle of attack of 5° was investigated in a water tunnel. The chord length was $c = 180$ mm and the chord-based Reynolds number was 25000. Cases with three free-stream conditions were examined: unobstructed flow with a transverse fluctuation intensity (in the free-stream at the wing-tip plane) $u_2'/U_\infty = 2.3\%$; "small-grid" turbulence with $u_2'/U_\infty = 3.5\%$ and a transverse integral length scale $L_2 = 0.063c$; and "large-grid" turbulence with $u_2'/U_\infty = 5.3\%$ and $L_2 = 0.078c$. Velocity maps were obtained on several transverse planes using stereo particle image velocimetry and three-dimensional, time-dependent vortex wandering was resolved using flow visualisation of fluorescent dye injected into the vortex at the wing tip. The results quantify the effect of turbulence on the amplitude, frequency and wavelength of the vortex wandering motion, as well as on the axial and azimuthal velocity variations within the vortex.

[1]Supported by NSERC

4:14PM L25.00004 Cutting, Splicing, and Kelvin Waves[1] , MARTIN SCHEELER, DUSTIN KLECKNER, WILLIAM T.M. IRVINE, University of Chicago — Recent experimental advances have allowed us to create, visualize and track vortices of prescribed shape and topology in classical fluids. We study the effect of surgery (cutting and splicing) on the evolution of the geometry and topology of these vortex loops, with a particular focus on the wave-like excitations generated by these operations. We interpret the dynamics of these excitations and the role they play within the broader context of vortex evolution.

[1]This work was supported by the National Science Foundation Materials Research and Engineering Centers (MRSEC) Program at the University of Chicago (DMR-0820054) and the Packard Foundation through a Packard fellowship.

4:27PM L25.00005 Three Dimensional Motions, Kelvin Waves, and Nanoparticle Tracking in Superfluid Helium , DAVID MEICHLE, DANIEL LATHROP, University of Maryland College Park — Liquid Helium becomes a quantum superfluid when cooled below the lambda transition temperature of 2.17 Kelvin. Superfluid helium exhibits interesting macroscopic effects such as zero viscosity; and its flow is irrotational except for the presence of line-like topological phase defects with quantized circulation called quantum vortices. The vortex dynamics can be observed by dispersing tracer particles into the fluid, which become trapped on the vortex cores. Using atmospheric ice particles our group recently observed the excitation and self-similar propagation of helical Kelvin waves on these quantized vortex cores following a vortex reconnection event. This observation of an intrinsically three dimensional phenomenon has motivated the development of a three dimensional imaging apparatus for liquid helium. We will present new data obtained by dispersing fluorescent nanoparticles tracers and our progress towards full three dimensional tracking of quantized vortex dynamics in liquid helium.

4:40PM L25.00006 Adaptive particle methods for barotropic vorticity dynamics on a rotating sphere[1] , PETER BOSLER, University of Michigan, LEI WANG, University of Wisconsin - Milwaukee, CHRISTIANE JABLONOWSKI, ROBERT KRASNY, University of Michigan — We present an adaptive particle method for barotropic vorticity dynamics on a rotating sphere. The flow map is represented by Lagrangian particles, organized into either triangular icosahedral panels or cubed-sphere quadrilateral panels. The particles carry vorticity and the panels are used to obtain quadrature weights in the point vortex approximation of the Biot-Savart integral. Adaptive panel refinement and remeshing are applied to maintain accuracy and reduce computational cost. Examples include Rossby-Haurwitz waves, Gaussian vortices, and perturbed jet dynamics.

[1]Supported by ONR

4:53PM L25.00007 Knotted Vortices: Entropic Lattice Boltzmann Method for Simulation of Vortex dynamics[1] , FABIAN BOESCH, SHYAM CHIKATAMARLA, ILYA KARLIN, ETH Zürich — Knotted and interlinked vortex structures in real fluids are conjectured to play a major role in hydrodynamic flow dissipation. Much interest lies in determining their temporal stability and the mechanism through which knots dissolve [1-3]. Kleckner and Irvine [1] recently have shown the existence of such knotted vortices experimentally by accelerating hydrofoils in water. In the present work we employ the entropic lattice Boltzmann method (ELBM) to perform DNS simulations of the creation and dynamics of knotted vortex rings inspired by the experimental setup in [1]. ELBM renders LBM scheme unconditionally stable by restoring the second law of thermodynamics (the Boltzmann H-theorem), and thus enables simulations of large domains and high Reynolds numbers with DNS quality [4-5]. The results presented in this talk provide an in-depth study of the dynamics of knotted vortices and vortex reconnection events and confirm the existence of trefoil knots in silicio for the first time.

[1] Klecker et al, Nature Physics, 9 (2013)
[2] Kida et al, Annu. Rev. Fluid Mech., 26 (1994)
[3] Ricca et al., J. Fluid Mech., 391 (1999)
[4] Karlin et al., Europhys. Lett. 47 (1999)
[5] Chikatamarla et al, Phys. Rev. Lett. 97, 010201 (2006)

[1]This work was supported by a grant from the Swiss National Supercomputing Centre (CSCS) under project ID s347

5:06PM L25.00008 Numerical simulation of the fluttering instability using a pseudospectral method with volume penalization , THOMAS ENGELS, M2P2-CNRS, Aix Marseille University, France & ISTA TU Berlin, Germany, DMITRY KOLOMENSKIY, McGill University / CRM, Montreal, Canada, KAI SCHNEIDER, M2P2-CNRS, Aix Marseille University, France, JOERN SESTERHENN, ISTA, TU Berlin, Germany — A new numerical scheme for the simulation of deformable objects immersed in a viscous incompressible fluid is presented. The 2d Navier-Stokes equations are discretized with a Fourier pseudo-spectral scheme. Using the volume penalization method arbitrary inflow conditions can be enforced, together with the no-slip conditions at the boundary of the immersed flexible object. The present work extends the penalization method to account for moving deformable objects while avoiding numerical oscillations in the hydrodynamic forces. For the solid part, a simple 1d model, the non-linear beam equation, is employed. The fluid and solid parts are coupled with a fast explicit staggered scheme. The fluttering instability of a slender structure immersed in a free stream is studied and three distinct states are obtained: stability of the initial condition or maintenance of an either periodic or chaotic fluttering motion. A detailed parameter study for different Reynolds numbers and reduced free-stream velocities is presented. The dynamics of the transition from a periodic to a chaotic state is investigated. The results are compared with those obtained by an inviscid vortex shedding method and by a viscous linear stability analysis, yielding for both satisfactory agreement.

5:19PM L25.00009 Formation and destabilization of Kelvin-Helmholtz billows in stably stratified turbulence , YOSHIFUMI KIMURA, Graduate School of Mathematics, Nagoya University, JACKSON HERRING, NCAR — We investigate the formation and destabilization of Kelvin-Helmholtz billows in stably stratified turbulence using the pseudo-spectral DNS of the Navier-Stokes equations under the Boussinesq approximation with 2048^3 grid points. Our method is to integrate the equations from the zero total energy initial condition with horizontal forcing imposed in a narrow wave number band. In the course of developments, the horizontal spectra first show a single steep power-law ($k^{74.5}$, where k is the horizontal wavenumber), and then the tail part of the spectrum begins to rise to show the Kolmogorov-type slope ($k^{75/3}$). From the viewpoint of vortex formation, we first observe that many wedge vortices are produced which move horizontally (like dipoles) in random directions. As time goes on, the wings of the wedges become thinner and thinner while translating, and finally detach to be almost independent vortex layers. This thinning mechanism makes the vertical shear stronger and eventually the local Richardson number small enough to produce Kelvin-Helmholtz billows. We will demonstrate that the transition in the horizontal energy spectra has a close relation with the destabilizing process of the Kelvin-Helmholtz billows.

5:32PM L25.00010 Regenerative growth due to axial flow induced by vortex-turbulence interaction , ERIC STOUT, FAZLE HUSSAIN, Texas Tech University — Direct numerical simulations of a vortex column embedded in fine scale homogeneous, isotropic turbulence reveals an inviscid mechanism for induction of axial flow on the column. Vortex threads, produced outside the column during vortex-turbulence interaction, are shown to drive the mechanism of axial flow generation. Oppositely oriented threads radially separate by self-induction, hence causing net axial flow. At computationally accessible Reynolds numbers (Re≡vortex circulation/viscosity=10 000), the axial flow due to a pair of oppositely signed vortex threads outside the column increases both with Re and time. At high Re, the axial flow can increase sufficiently to render the vortex column unstable by the well-known q criterion. The vorticity field reveals that axial flow is another mechanism, perhaps more dominant than the parent-offspring hairpin vortex scenario (Hussain, Pradeep & Stout JFM 2011), of regenerative energy growth – likely to be important for implementing breakup of aircraft trailing vortices.

Monday, November 25, 2013 3:35PM - 5:32PM –

Session L26 Reacting Flows VII: Experiments 321 - Bruce Chehroudi, Advanced Technology Consultants

3:35PM L26.00001 Experimental investigation of laboratory fire whirls , KATHERINE HARTL, Princeton University, ALEXANDER SMITS, Princeton University, Monash University — A fire whirl is a swirling diffusion flame that may occur to great destructive effect in urban fires or wildfires. To study fire whirls in the laboratory, we use a burner flame, using DME as fuel, and induce the swirl by entraining air through a split cylinder surrounding the central flame. Stereo Particle Image Velocimetry (PIV) is used to obtain distributions of the three components of velocity outside the fire whirl core. The effects of fuel flow rate, gap width, and height along the flame are examined, and the scaling behavior is investigated.

3:48PM L26.00002 Laser-Diagnostic Mapping of Temperature and Soot Statistics in a 2-m Diameter Turbulent Pool Fire , SEAN KEARNEY, Sandia National Laboratories — We present spatial profiles of temperature and soot-volume-fraction statistics from a sooting 2-m base diameter turbulent pool fire. Dual-pump coherent anti-Stokes Raman scattering (CARS) and laser-induced incandescence (LII) are utilized to obtain profiles of temperature and soot probability density functions (pdf) at three vertical heights above the surface of the methanol/toluene fuel pool. The experiments are conducted in the unique Sandia FLAME facility, which has recently been modified to allow for vertical translation of the optical systems and horizontal translation of the liquid fuel burner. Results are presented both in the fuel vapor-dome region at 0.25 base diameter and in the actively burning region at 0.5 and 0.75 diameters above the fuel surface. The evolution of the soot and temperature pdfs is discussed, profiles of the temperature and soot mean and rms statistics are provided, and initial estimates of the joint temperature/soot statistics, which describe soot radiative emission, are presented.

4:01PM L26.00003 Experimental investigation of boundary-layer flashback in swirl flames , DOMINIK EBI, NOEL CLEMENS, The University of Texas at Austin — Swirling flows are widely employed for flame stabilization purposes in gas turbine combustors. However, flames in swirling flows are more prone to flashback, a potentially catastrophic phenomenon leading to thermal damage of the burner. The physical mechanism driving flashback in a swirling flow is not yet fully understood. The mechanism is particularly complex if the upstream flame propagation interacts with a boundary layer. In a previous study we showed that the flame/boundary-layer interaction is important for burners, which include an axial swirler and a central body attached to the swirler hub. We are investigating the mechanism of flashback in atmospheric pressure lean-premixed methane/hydrogen-air flames inside the mixing tube of our confined model swirl combustor. Flashback occurs at an equivalence ratio of approximately 0.7. The effect of hydrogen is investigated by testing different methane-to-hydrogen ratios. The duration of a single flashback event is on the order of 100 ms, requiring high-speed diagnostic techniques. We are applying simultaneous stereoscopic PIV, flame front detection based on Mie scattering, and chemiluminescence imaging to investigate the flame/flow interaction during flashback events.

4:14PM L26.00004 Oscillatory Flame Response in Acoustically Driven Fuel Droplet Combustion[1] , BRETT LOPEZ, CRISTHIAN SEVILLA, TAKESHI SHOJI, ARI EKMEKJI, OWEN SMITH, ANN KARAGOZIAN, University of California, Los Angeles — This experimental study focuses on droplet combustion characteristics for various liquid fuels during exposure to external acoustical perturbations generated within an acoustic waveguide. The study examines combustion during excitation conditions in which the droplet is situated in the vicinity of a pressure node (PN). In response to such acoustic excitation, the flame surrounding the droplet is deflected, on average, with an orientation depending on the droplet's relative position with respect to the PN. Flame orientation is always found to be consistent with the sign of a theoretical bulk acoustic acceleration, analogous to a gravitational acceleration.[2] Yet experimentally measured acoustic accelerations based on mean flame deflection differ quantitatively from that predicted by the theory. Phase-locked OH* chemiluminescence imaging reveals temporal oscillations in flame standoff distance from the droplet as well as chemiluminescent intensity which are especially pronounced when the droplet is situated close to the PN. Quantification of combustion-acoustic coupling via the Rayleigh index reveals a more detailed understanding of dynamical phenomena.

[1]Supported by the Air Force Research Lab/ERC Inc. and the UC CARE and MSD Scholars programs.
[2]Tanabe, et al., PCI, 2000

4:27PM L26.00005 Similarity and Scaling of Turbulent Flame Speeds for Expanding Premixed Flames of C_4-$C_8$$n$-alkanes[1] , FUJIA WU, ABHISHEK SAHA, SWETAPROVO CHAUDHURI, SHENG YANG, CHUNG K. LAW, Princeton University — We experimentally investigated the propagation speed of constant-pressure expanding flames in near isotropic turbulence using a dual-chamber, fan-stirred vessel. The motivation is to test whether the fuel similarity concept among C_4-C_8 n-alkanes on laminar flames also holds for turbulent flames. Previously it was found that the laminar flame speed and Markstein length are almost identical for C_4-C_8 n-alkanes. If this fuel similarity concept can also be shown for turbulent flames, it will suggest a canonical flame structure for large hydrocarbon fuels, *i.e.*, large fuels always decompose to small C_0-C_4 fuel fragments before being oxidized, and would significantly simplify the description of the flames. Preliminary results show that in the flamelet and thin-reaction zone, turbulent flame speeds of C_4-C_8 n-alkanes are indeed largely similar at various conditions, thereby suggesting the fuel similarity for turbulent flames. In addition, it is found that the normalized turbulent flame speed also approximately scales with the square root of an appropriately-defined Reynolds number recently found for C_0-C_4 fuels.

[1]This work was supported by the AFOSR under the technical monitoring of Dr. Chiping Li.

4:40PM L26.00006 Structure and Dynamics of a Reacting Jet in a Swirling Vitiated Crossflow[1] , PRATIKASH PANDA[2], MARIO ROA[3], ROBERT LUCHT[4], Purdue University, West Lafayette — A reacting jet issuing into a vitiated, swirled cross flow operating at a pressure of 5.5 bars is investigated using high repetition rate (5 kHz) Particle Image Velocimetry (PIV). A premixed jet composed of natural gas and air is injected into the vitiated stream through an extended nozzle downstream of a low swirl burner (LSB) that produces the vitiated, swirled flow. The jet-to-crossflow momentum flux ratio is varied to study the corresponding impact on the flow structures. The raw PIV images indicate a strong influence of the swirling crossflow on the jet at planes closest and farthest away from the nozzle exit. The mean flow field is found to preserve the counter-rotating vortex pair (CRVP) which is the most dominant flow structure in a jet in crossflow flow-field. The instantaneous vector field indicates presence of a very interesting oscillatory motion of the CRVP indicating a in plane jet flapping behavior. This oscillatory motion is affected by the accelerating flow above and below the CRVP which could be correlated to the instantaneous heat release at that location. Proper Orthogonal Decomposition (POD) is used to extract the frequency characteristics and dynamics of this flapping jet.

[1]Funding support from Siemens and DOE UTSR.
[2]Graduate Research Assistant
[3]Graduate Research Assistant
[4]Professor, School of Mechanical Engineering

4:53PM L26.00007 Construction and Characterization of a Shock Tube for Ignition and Pollutant Formation Studies , CORY PRYKULL, ROBERT DREIKER, Department of Mechanical and Aerospace Engineering, Syracuse University, MARCOS FERNANDES, Fundacao Educacional Inaciana, Sao Bernardo do Campo, Brazil, MAZEN ELDEEB, BEN AKIH-KUMGEH, Department of Mechanical and Aerospace Engineering, Syracuse University — Shock tubes are versatile research facilities with wide applications in aerodynamics, high-temperature chemical kinetics and medical research. We discuss the construction and the gas dynamics characterization of such a facility for combustion studies with a focus on ignition and pollutant formation. Measures to achieve high quality post-reflected shock conditions with minimal shock-boundary layer interactions are discussed. Characterization of the facility is first carried out using non-reactive gases in order to assess the quality of the post-reflected shock conditions and the available test times. The incident velocity is determined using fast response pressure transducers. Experimentally observed post-reflected shock pressure profiles are compared with predictions of one dimensional shock equations, which also allow for the calculation of temperature. Subsequent shock tube ignition experiments are carried out for selected fuel and oxidizer systems from the literature in order to validate and thereby, demonstrate the suitability of the facility for combustion studies. Further measurements of soot volume fractions under fuel rich conditions are realized by means of laser extinction.

5:06PM L26.00008 ABSTRACT WITHDRAWN –

5:19PM L26.00009 Schlieren Imaging of Chemically-Induced Flow Instabilities During Step-Growth Polymerization , PATRICK BUNTON, MICHAEL RAWAT, SIMONE STEWART, Department of Physics, William Jewell College, ANNE DE WIT, Nonlinear Physical Chemistry Unit, Universit Libre de Bruxelles, Bruxelles, Belgium, JOHN POJMAN, Department of Chemistry, Louisiana State University, Baton Rouge, LA, USA — Schlieren imaging was used to observe the dependence on degree of reactivity on flow instabilities during step-growth polymerization. For example a solution of a 2,2'-(ethylenedioxy)- diethanethiol containing varying concentrations of Octylamine as initiator was used to displace a trimethylolpropane triacrylate monomer. The concentration of initiator was used to control the degree of reactivity of the solutions and therefore the Damkohler Number which is the ratio of the chemical to the hydrodynamic timescale. By varying the concentration of initiator and/or the functionality of the monomers, one can tune the reaction product from a viscous liquid, to gel, or even a solid. Resulting flow patterns were found to depend on degree of reactivity and effects were observed regardless of the direction of flow. Also observed was evidence of three-dimensional effects on the resultant flow patterns.

Monday, November 25, 2013 3:35PM - 5:19PM –

Session L28 Biofluids: General VI - Fluid Film Flows

Spirit of Pittsburgh Ballroom B/C - Amir Hirsa, Rensselaer Polytechnic Institute

3:35PM L28.00001 Tear Movement through a Contact Lens of Variable Thickness , MATTHEW GERHART, DANIEL ANDERSON, George Mason University — This work is on a two-dimensional tear film with a movable porous contact lens. The inclusion of a contact lens into a tear film results in three layers: Pre-Lens Tear Film, Contact Lens, and the Post-Lens Tear Film layers. The interfaces between the contact lens and the tear films are modeled as planar interfaces. There is a free surface interface between the tear film and the outside air. The goal is analyze the effects of the spatial variability of thickness on the Post-Lens Tear Film thickness and on the fluid flow through the Contact Lens layer.

3:48PM L28.00002 The influence of surfactant on the stability of a liquid bilayer inside a rigid tube[1] , YUANYUAN SONG, DAVID HALPERN, University of Alabama, JAMES GROTBERG, University of Michigan — Many airways in the lung are coated with a bilayer consisting of a serous layer adjacent to a more viscous mucus layer which is contiguous with the gas core. An instability due to high surface tension at the interfaces may lead to the formation of a liquid plug that blocks the passage of air. This phenomenon is known as airway closure. Here we investigate the linear stability for the case when the thin liquid bilayer is Newtonian and coated within a rigid tube with the presence of an insoluble surfactant monolayer at the mucus-gas interface. Surfactant affects the surface tension and also induces a surface stress at the interface. A system of equations for the deflections of the interfaces and the surfactant concentration is derived by using lubrication theory. These equations are linearized, and by applying the method of normal modes, a dispersion equation for the growth rate of the disturbances is obtained. Its dependence on the viscosity ratio, the thickness ratio of the two liquid layers, the base state surface tension ratio, and the Marangoni number is investigated, and comparisons with previous single layer models are discussed.

[1]Supported by NIH grant HL085156.

4:01PM L28.00003 A Model for Lipid Layer Dynamics on a Moving Domain , NICHOLAS GEWECKE, RICH BRAUN, University of Delaware, CHRIS BREWARD, University of Oxford — The human tear film consists of an aqueous layer and a thinner layer of nonpolar lipids, with polar lipids along the interface between them. Dynamics of the nonpolar lipid layer are not yet well understood. Experimental observations indicate that visible features in the lipid layer can persist through multiple blinks, but how this occurs is a matter of debate. One possibility is a concertina-like folding, corresponding to a serpentine instability in the lipid layer during a blink. Another possibility is due to a varicose instability. We use a two-layer thin film model to understand the dynamics of the lipid layer. We study the dynamics of a model Newtonian extensional layer floating on a less viscous shear layer. The upper layer includes van der Waals terms so that it dewets from the lower layer as expected in the tear film system. We give results for expanding and contracting domains for both varicose and sinuous disturbances.

4:14PM L28.00004 A New Model for the Suction Pressure Under the Contact Lens[1] , KARA MAKI, DAVID ROSS, EMILY HOLZ, Rochester Institute of Technology — We study the dynamics of the contact lens to better understand how the design of the lens can be optimized for patient comfort and ocular fit. When a contact lens is inserted on an eye, it is subjected to forces from both the tear film in which it is immersed and the blinking eyelid. In response, the lens bends and stretches. These forces center the lens, and they produce the suction pressure that keeps the lens on the cornea. In this presentation, we couple fluid and solid mechanics to determine the most prominent forces acting on the lens. We present a mathematical model that predicts the suction pressure. We explore the influence of contact lens properties on the suction pressure.

[1]This work is supported by the Economic Development Administration and Bausch + Lomb.

4:27PM L28.00005 How flies clean their eyes , GUILLERMO AMADOR, FABIEN DURAND, WENBIN MAO, ALEXANDER ALEXEEV, DAVID HU, Georgia Institute of Technology — Flying insects face a barrage of foreign particles such as dust and pollen, which threaten to coat the insect's eyes and antennae, limiting their sensing abilities. In this study, we elucidate novel aerodynamic and elastic mechanisms by which insects keep these organs clean. The compound eye of many species of insects is covered by an array of short bristles, or setae, evenly spaced between each photoreceptor unit. Among these insect species, setae length is triple their spacing. We conduct numerical simulations and wind tunnel experiments using an insect eye mimic to show this critical setae length reduces shear rate at the eye surface by 80%. Thus, the setae create a stagnant zone in front of the eye, which diverts airflow to reduce deposition of particles. Setae can also act as springboards to catapult accumulated particles. In high speed videography of insects using their legs to clean themselves, we observe deflected setae hurling micron scale particles at accelerations over 100 times earth's gravity. The dual abilities of setae to divert airflow and catapult particles may motivate bio-inspired designs for dust-controlling lenses, sensors, and solar panels.

4:40PM L28.00006 Amyloid fibril formation at a uniformly sheared air/water interface , DAVID POSADA, AMIR HIRSA, Rensselaer Polytechnic Institute — Amyloid fibril formation is a process by which protein molecules in solution form nuclei and aggregate into fibrils. Amyloid fibrils have long been associated with several common diseases such as Parkinson's disease and Alzheimer's. More recently, fibril protein deposition has been implicated in uncommon disorders leading to the failure of various organs including the kidneys, heart, and liver. Fibrillization can also play a detrimental role in biotherapeutic production. Results from previous studies show that a hydrophobic interface, such air/water, can accelerate fibrillization. Studies also show that agitation accelerates fibrillization. When attempting to elucidate fundamental mechanisms of fibrillization and distinguish the effects of interfaces and flow, it can be helpful to experiment with uniformly sheared interfaces. A new Taylor-Couette device is introduced for in situ, real-time high resolution microscopy. With a sub-millimeter annular gap, surface tension acts as the channel floor, permitting a stable meniscus to be placed arbitrarily close to a microscope to study amyloid fibril formation over long periods.

4:53PM L28.00007 Dynamics of the Primary Cilium in an Oscillatory Flow , YUAN-NAN YOUNG, New Jersey Institute of Technology — In this work we investigate the dynamics of a primary cilium under oscillatory flows. The primary cilium is modeled as an elastic slender beam coupled to an elastic shell with a local torque (mimicking the sub-axonemal anchorage) at the beam-shell junction. We examine how a primary cilium responds to oscillatory flows depending on its axonemal stiffness and the initial base angle. In particular we focus on the tension and forces at the cilium base where ion channels are speculated to be "activated" by fluid flow via cilium bending. We find that a tilted cilium base gives rise to slightly larger magnitude in tension and forces at the base. We further compare the cilium bending dynamics between oscillating and pulsing flows, and investigate the effect of oscillation frequency. From our simulation results we speculate that the reduced ability of periodic pulsing flow to stimulate the primary cilia responses at high frequencies may be due to lack of time for ion channels to respond to the stress at the filament base.

5:06PM L28.00008 The influence of nonpolar lipids on tear film dynamics[1] , CHRIS BREWARD, University of Oxford — We will examine the effects of the presence of nonpolar lipids on the evolution of a tear film during a blink. We will track the thickness of the aqueous tear layer, the thickness of the nonpolar lipid layer, and the concentration of the polar lipids that reside between the two. Our model can be reduced in various limits to previous models for tear dynamics studied. We present numerical solutions for the evolution of the tear film and show how the key parameters play a role in determining how the nonpolar lipid spreads.

[1]This work was partially supported by Award No. KUK-C1-013-04 made by King Abdullah University of Science and Technology (KAUST).

Monday, November 25, 2013 3:35PM - 5:45PM –

Session L30 Instability: Rayleigh-Taylor II 408 - Daniel Livescu, Los Alamos National Laboratory

3:35PM L30.00001 Comparison Between Turbulence Model Initialization Approaches for Rayleigh-Taylor and Richtmyer-Meshkov Turbulent Mixing , BERTRAND ROLLIN, NICK DENISSEN, JON REISNER, MALCOLM ANDREWS, Los Alamos National Laboratory — Implementation and validation cases of a novel approach to initialization for RANS simulations of interfacial instability induced by mixing are presented. The strategy consists of using an analytical model to compute the instability evolution from the quiescent state, and make use of its prediction to generate initial conditions for the turbulence model. Explicitly, an incompressible inviscid model for Rayleigh-Taylor and Richtmyer-Meshkov instabilities continuously updates the turbulence model variables values in the mixing layer, until the Reynolds number suggests that the flow has become turbulent. Implementation of this procedure is made in three steps: first, the instability model is run alone while the interface is evolved by the hydrocode hosting the turbulence model; second, the turbulence model is started and the turbulence variables updated in accordance with the instability growth model prediction; finally, the Reynolds number suggests that the turbulent mixing regime is reached, causing the instability model to stop and the turbulence model to continue alone. The initialization methodology is applied to Rayleigh-Taylor and Richtmyer-Meshkov problems. Comparisons between simulations using a traditional initialization technique and the new initialization approach are presented and discussed.

3:48PM L30.00002 Blast-Driven Hydrodynamic Instability[1] , MARC T. HENRY DE FRAHAN, ERIC JOHNSEN, University of Michigan — Accurate characterization of mixing from hydrodynamic instabilities, such as Richtmyer-Meshkov, Rayleigh-Taylor, and Kelvin-Helmholtz, is important to many multi-fluid applications, particularly, inertial confinement fusion, supernova collapse, and scramjet combustion. We investigate the dynamics of a perturbed interface between two fluids subjected to a planar blast wave. An initial point source explosion initiates a blast, which can be described as a shock front followed by a rarefaction wave. The interface, therefore, experiences an instantaneous acceleration (a pressure increase) followed by a gradual, time-dependent deceleration (a pressure decrease). The resulting interaction gives rise to Richtmyer-Meshkov and Rayleigh-Taylor growth, depending on the shock strength and blast profile. Using a high-order accurate numerical method that prevents pressure errors at interfaces when simulating variable specific heats ratios, we identify regimes in which one or the other instability dominates.

[1]This research was supported by the DOE NNSA/ASC under the predictive Science Academic Alliance Program by Grant No. DEFC52-08NA28616.

4:01PM L30.00003 Temporal Evolution and Scaling of Mixing in Two-dimensional Rayleigh-Taylor Turbulence[1] , QUAN ZHOU, Shanghai Institute of Applied Mathematics and Mechanics, Shanghai University, Shanghai 200072, China, QUAN ZHOU TEAM — We report a high-resolution numerical study of two-dimensional (2D) miscible Rayleigh-Taylor (RT) incompressible turbulence with the Boussinesq approximation. We present results from an ensemble of 100 independent realizations performed at unit Prandtl number and small Atwood number with a spatial resolution of 2048×8193 grid points and Rayleigh number up to $Ra \sim 10^{11}$. Our main focus is on the temporal evolution and the scaling behavior of global quantities and of small-scale turbulence properties. Our results show that the buoyancy force balances the inertial force at all scales below the integral length scale and thus validate the basic force-balance assumption of the Bolgiano-Obukhov scenario in 2D RT turbulence. It is further found that the Kolmogorov dissipation scale $\eta(t) \sim t^{1/8}$, the kinetic-energy dissipation rate $\varepsilon_u(t) \sim t^{-1/2}$, and the thermal dissipation rate $\varepsilon_\theta(t) \sim t^{-1}$. All of these scaling properties are in excellent agreement with the theoretical predictions of the Chertkov model [Phys. Rev. Lett. 91, 115001 (2003)].

[1]This work was supported by the Natural Science Foundation of China (NSFC) under Grant Nos. 11222222 and 11161160554 and Innovation Program of Shanghai Municipal Education Commission under Grant No. 13YZ008.

4:14PM L30.00004 Electrohydrodynamically induced mixing in immiscible multilayer flows , RADU CIMPEANU, DEMETRIOS PAPAGEORGIOU, Imperial College London — In this study we investigate electrostatic stabilization mechanisms acting on stratified fluids. A classical example shows how an electric field can be used to control and even suppress the Rayleigh-Taylor instability when a heavy fluid lies above lighter fluid. We present a linear stability study, as well as extensive direct numerical simulations via the volume of fluid method to show that when the fluids are dielectrics and an electric field acts horizontally (in the plane of the undisturbed liquid-liquid surface), growth rates and critical stability wavenumbers are reduced thus shifting the instability to longer wavelengths. Agreement between linear theory and direct numerical simulations is shown to be excellent. From a practical perspective, we aim to identify active control protocols in confined geometries that induce time dependent flows in small scale devices without having moving parts. This effect has numerous applications, ranging from mixing phenomena to electric lithography. Two- and three-dimensional computations are carried out and several such protocols are described.

4:27PM L30.00005 The Rayleigh-Taylor Instability driven by an accel-decel-accel profile[1] , PRAVEEN RAMAPRABHU, VARAD KARKHANIS, University of North Carolina at Charlotte, ANDREW LAWRIE, University of Bristol, United Kingdom — We describe numerical simulations of the miscible Rayleigh-Taylor (RT) instability driven by a complex acceleration history, g(t), with initially destabilizing acceleration, $g > 0$, an intermediate stage of stabilizing deceleration, $g < 0$, and subsequent destabilizing acceleration, $g > 0$. Initial perturbations with both single wave-number and a spectrum of wave-numbers (leading to a turbulent front) have been considered with these acceleration histories. We find in the single-mode case that the instability undergoes a so-called phase inversion during the first acceleration reversal from $g > 0$ to $g < 0$. If the zero-crossing of g(t) occurs once the instability growth has reached a state of nonlinear saturation, then hitherto rising bubbles and falling spikes reverse direction and collide, resulting in small-scale structures. For multi-mode perturbations, we find that bubbles and spikes collide during phase inversion, the interfacial region is turbulent, and undergoes a period of enhanced structural breakdown. This is accompanied by a rapid increase in the rate of molecular mixing, and increasing isotropy within the region. During the final stage of $g > 0$ acceleration, self-similar RT mixing re-emerges, together with a return to anisotropy.

[1]This work was supported in part by the (U.S.) Department of Energy (DOE) under Contract No. DE-AC52-06NA2-5396.

4:40PM L30.00006 Initial Condition Effects on Turbulent Rayleigh Taylor Instability under Variable Acceleration History , DENIS ASLANGIL, Lehigh University, ANDREW LAWRIE, University of Bristol, ARINDAM BANERJEE, Lehigh University — Initial condition effects on Rayleigh Taylor Instability are investigated for various fixed and variable acceleration histories. A massively parallel high resolution code (MOBILE) is used to model incompressible flow using an Implicit Large Eddy simulation technique. The simulations are initialized to understand the effects of spectral index, spectral bandwidth and discrete banded spectra. This study will present both general low-order metrics such as mix widths and growth constants and will compare the results for different acceleration histories and initial conditions. Studies on higher order turbulence parameters such as second order moments, their dissipations, and production–dissipation ratios will also be presented and are important to identify the similarities and differences between the Rayleigh–Taylor turbulence and the more conventional stationary turbulence.

4:53PM L30.00007 ABSTRACT WITHDRAWN –

5:06PM L30.00008 ABSTRACT WITHDRAWN –

5:19PM L30.00009 Mixing and turbulence generated by the tilted Rayleigh-Taylor instability , DANIEL LIVESCU, Los Alamos National Laboratory, TIE WEI, New Mexico Tech — In most practical applications of Rayleigh-Taylor instability (RTI), the initial interface is not perpendicular to the direction of acceleration. When the degree of tilting of the interface is non-negligible, the resulting mean flow is no longer one-dimensional as is the case with the classical RTI, and the two main turbulence production mechanisms, buoyancy and shear, are both present. The development of the instability can be decomposed into a large overturning motion, which leads to a strengthening of the mean shear, the formation of a large side wall bubble and spike, and the interior mixing layer growth. Results from very large Direct Numerical Simulations are presented of this unique unit problem and used to study the competition between shear and buoyancy production of turbulence and the respective effects on the mixing and turbulence properties. In particular, the development of the mixing layer seems more sensitive to the properties of the initial perturbation than in classical RTI.

5:32PM L30.00010 The Rayleigh-Taylor instability for a thin film on the inside of a horizontal cylinder , NAIMA HAMMOUD, Princeton University, PHILIPPE TRINH, PETER HOWELL, JONATHAN CHAPMAN, Oxford University, HOWARD STONE, Princeton University — Thin films on curved surfaces are widely observed in coating and painting processes and wetting problems. We consider a thin film on a curved substrate under the effect of gravitational, viscous, and surface tension forces. When the film is on the underside of the substrate, gravity works as a destabilizing force, and a Rayleigh-Taylor type instability is expected. We consider the stability of a uniform thin film coating the inside of a horizontal circular cylinder. Using asymptotic methods, we find that instabilities are of a transient nature, thus showing that curvature helps stabilize the film. We also find that these "instabilities" occur primarily in the angular direction with the axial perturbations only appearing as higher-order corrections. These results seem to agree well with experiments (H. Kim et al., this conference).

Monday, November 25, 2013 3:35PM - 5:45PM –

Session L31 Free-Surface Flows III 402 - John McHugh, University of New Hampshire

3:35PM L31.00001 Experimentally observed flows inside inkjet-printed aqueous rivulets , VADIM BROMBERG, TIMOTHY SINGLER, SUNY Binghamton — Understanding the flow inside sessile liquid masses of different shapes is important in a variety of solution-based material deposition and patterning processes. We investigated the shape evolution and internal flow of inkjet-printed aqueous rivulets of finite length using optical microscopy. Rivulets were formed by printing a pre-determined number of drops at controlled frequency and spatial overlap. Capillary-driven rivulet breakup into individual drops was inhibited by chemical modification of substrates that resulted in controlled contact angle hysteresis with zero static receding contact angle. A variety of novel capillary- and evaporatively-driven flows were identified using fluorescent particles as flow tracers. Flow regimes were investigated as a function of advancing contact angle, deposition parameters, and fluid properties.

3:48PM L31.00002 Investigation of bubble entrainment by breaking waves in turbulent two-phase Couette flows[1] , DOKYUN KIM, ALI MANI, PARVIZ MOIN, CTR, Stanford University — The bubbles entrained by breaking waves have an important role in understanding the upper-ocean physical processes. Although the size distribution of bubbles is most important in these processes, its characteristics have not been clearly understood because measurement of the bubble size is challenging especially in the high void-fraction region. In the present study, numerical simulations are performed to investigate the bubble formation mechanism in breaking waves of turbulent two-phase flow with moving side-walls. A newly developed conservative VOF method[2] coupled to a subgrid Lagrangian breakup model is used to resolve wave breakup phenomenon and resulting bubbles. The numerical method is tested and validated against the experiments for canonical problems. The Reynolds and Froude numbers considered are 12,760 and 6.8, respectively. In order to investigate the effect of Weber number on the characteristics of the bubble size, the simulations are conducted for two different Weber numbers. The statistics and scale properties of bubbles will be presented and discussed.

[1]Supported by the Office of Naval Research
[2]Developed at Cascade Technologies

4:01PM L31.00003 Free-surface flow driven by a deforming boundary , FREDERIK BRASZ, Princeton University, JOHN LISTER, ITG, DAMTP, University of Cambridge, CRAIG ARNOLD, Princeton University, ARNOLD GROUP COLLABORATION — We present an analytical solution for flow in a liquid layer driven by a deforming boundary. An initially flat wall undergoes a sinusoidal deformation with small amplitude relative to wavelength, imparting momentum to the fluid. Initially, the flow is directed away from the crests and slows with the slowing of the boundary motion. A domain perturbation method is used to reveal that even when the boundary stops moving, nonlinear interactions with the free surface leave a remnant momentum, and this momentum is directed back toward the crests, a precursor to jet formation. This scenario arises in a laser-induced printing technique in which an expanding blister imparts momentum into a liquid film to form a jet. This analysis provides insight into the physics underlying interactions between deforming boundaries and free surfaces, in particular the dependence on the thickness of the liquid layer relative to the deformation wavelength. Numerical simulations are used to verify the theory and show its range of validity.

4:14PM L31.00004 Numerical simulation of a turbulent wall jet in a rough-bed open channel[1] , JOONGCHEOL PAIK, Gangneung-Wonju National University, FABIAN BOMBARDELLI, KEN LOH, University of California, Davis — Numerical results of the mean flow and turbulence characteristics in the near field of a turbulent plane wall jet issuing from a sluice gate onto a rough flat wall are presented. The flow had been experimentally investigated by Albayrak et al. [I. Albayrak, E.J. Hopfinger and U. Lemmin, J. Fluid Mech., 606, 27, (2008)] at the Reynolds number of 33,500 and the Froude number of 1.014, based on the jet velocity and the sluice gate opening height. Turbulent flow is simulated using the k-ω shear-stress transport (SST) model and the scale-adapted simulation (SAS) based on the SST model. The jet velocity profile is numerically reproduced based on the difference of upstream and downstream water levels computed by the volume of fluid method. The numerical results show that the outlet boundary should be carefully treated to successfully reproduce the velocity profile approaching the shape of the typical open-channel flow downstream of the attachment point of the jet. Numerical solutions appear to agree reasonably well with the measurement in terms of the outer-layer spatial growth rate and Reynolds stress distributions. The mesh convergence of numerical solutions is also presented.

[1]Supported by the NSF

4:27PM L31.00005 Dispersive Hydrodynamics in Viscous Fluid Conduits , NICHOLAS LOWMAN, MARK HOEFER, Department of Mathematics, North Carolina State University — The evolution of the interface separating a conduit of light, viscous fluid rising through a heavy, more viscous, exterior fluid at small Reynolds numbers is governed by the interplay between buoyancy and viscous stress. Perturbations about a state of vertically uniform, laminar conduit flow are considered in the context of the Navier-Stokes equations with appropriate boundary conditions, which lead systematically to a maximal balance between buoyancy driven, nonlinear self-steepening and viscous, interfacial stress induced, nonlinear dispersion. This results in a scalar, nonlinear partial differential equation describing large amplitude dynamics of the cross-sectional area of the intrusive fluid conduit. Unsteady perturbations of the uniform state have been shown in a laboratory setting to produce hallmark features of nonlinear, dispersive systems including solitary waves and nonlinear wave trains, i.e. dispersively regularized shock waves (DSWs). Shock waves solutions to the conduit equation for step-like initial data exhibit novel DSW behaviors, including backflow and DSW implosion. The asymptotic analysis shows that these fully nonlinear, dispersive hydrodynamic features of the reduced model are experimentally accessible in viscous fluid conduits.

4:40PM L31.00006 Free surface waves on a horizontal shear , GARY LAPHAM, Maine Maritime Academy, JOHN MCHUGH, University of New Hampshire — Free surface waves on a non-uniform mean flow are considered. The mean flow $U(y)$ varies with the transverse coordinate y but not the vertical. The domain is bounded on one side by a flat rigid vertical wall and unbounded on the other side. The mean flows considered are nonzero near the vertical wall and approach zero far from the wall, e.g. $U = e^{-\gamma y}$. For large y where the mean flow is near-zero the waves are merely irrotational Stokes' waves. Near the wall the mean flow and the waves are rotational but still inviscid. Linear solutions are obtained for several canonical cases with a nonuniform coordinate transformation that converts the free surface boundary condition into a Bessel equation. A Bessel expansion provides the velocity components, then wavespeeds are obtained numerically. Steady waves are found with wavespeeds outside the range of U, matching previous results in a flow bounded on both sides.

4:53PM L31.00007 CFD Experiments for Wind-Turbine-Platform Seakeeping Models and Flow Physics , ALEXANDER DUNBAR, Penn State University, ERIC PATERSON, Virginia Tech University, BRENT CRAVEN, JAMES BRASSEUR, Penn State University — As part of the Penn State "Cyber Wind Facility," we describe the development and application of a tightly-coupled CFD/6-DOF solver in OpenFOAM for the simulation of offshore floating wind turbine platforms. We highlight the tightly-coupled computational framework and validation of the solver via a comparison with benchmark experimental measurements. The validated CFD/6-DOF solver is then applied to the OC4 DeepCwind semisubmersible for the prediction of platform motion due to wind and wave loading. Supported by the US Department of Energy.

5:06PM L31.00008 Flow past a cylinder near a free surface[1] , KEEGAN DELANEY, MARCOS VANELLA, ELIAS BALARAS, The George Washington University, AMIR RIAZ, University of Maryland — Flow past a cylinder close to a free surface gives rise flow phenomena that are very different from ones in cases where the cylinder is fully submerged. In this study we will report resolved large-eddy simulations, where we examine the effects of various parameters that have been shown to effect the flow phenomena in previous experimental studies. In all computations a Navier-Stokes solver for multiphase incompressible flows with immersed boundaries and Adaptive Mesh Refinement is utilized. It employs level-set techniques to sharply define the interface between different phases. A fractional step method is used to solve the momentum and continuity equations, which results in a variable coefficient Poisson pressure equation. Proper jump conditions are applied to the Poisson pressure equation to accurately capture the jump in pressure that results from surface tension between different phases. Scalability and efficiency were placed at a premium during development of the solver, which has been tested to core counts on the order of 10,000. We will present details on the interactions between the free surface and vortices shed from the cylinder and their impact in the structure of the immediate wake and air entrainment.

[1]Supported by ONR N000141110588 monitored by Dr. Ki-Han Kim

5:19PM L31.00009 Universal Froude number in a circular hydraulic jump, implication on the jump radius selection[1] , ALEXIS DUCHESNE, LUC LEBON, LAURENT LIMAT, Laboratoire Matière et Systèmes Complexes - CNRS - Université Paris Diderot — In the literature, it is known that the properties of a standard hydraulic jump depend critically on a Froude number Fr defined by the ratio between the flow speed and the gravity waves speed: Fr is larger than 1 upstream of the shock, and smaller than 1 downstream, an accumulation of gravity waves occuring at the shock with formation of a sharp liquid wall. Surprisingly, to our knowledge, this question of Froude number value has never been explored for the circular hydraulic jump formed by an impinging jet on a horizontal surface. We have investigated carefully this question, varying the flow rate, the liquid viscosity and the surface tension. We have found that, in the specific case of a circular jump with no confinement walls, there exists an universal value, equal to 0.38 on which the Froude number defined at the jump exit is locked. We examine the implications of this result on the selection of the jump radius R, after combining it with the large scale flow structure around the jump, calculated in the lubrication limit. In agreement with our data, R is very close to follow the law proposed by Bohr, but this law has to be modified by introducing non negligible logarithmic corrections.We also discuss the implications of our results in terms of Watson description of the shock.

[1]This work was sponsored by the French National Agency for Research.

5:32PM L31.00010 SPH Simulation of Liquid Scattering from the Edge of a Rotary Atomizer , SEIICHIRO IZAWA, TAKUYA ITO, MASAYA SHIGETA, YU FUKUNISHI, Tohoku University — Three-dimensional incompressible SPH method is used to simulate the behavior of liquid scattering from the edge of a rotary atomizer. Rotary atomizers have been widely used for spraying, painting and coating, for instance, in the automobile industry. However, how the spray droplets are formed after leaving the edge of the rotary atomizer is not well understood, because the scale of the phenomenon is very small and the speed of rotation is very fast. The present computational result shows that while the liquid forms a film on the surface of the rotating disk of the atomizer, it quickly deforms into many thin columns after leaving the disk edge, and these columns soon break up into fine droplets which spread out in the radial direction. The size of droplets tends to become smaller with the increase in the disk rotating speed. The results show good agreement with the experimental observations.

Monday, November 25, 2013 3:35PM - 5:45PM –

Session L32 Geophysical: General I - Rotating Flows 403 - Jean-Marc Chomaz, LadHyX, CNRS-Ecole Polytechnique

3:35PM L32.00001 Dynamically Consistent Shallow-Atmosphere Equations with a Complete Coriolis force[1] , MARINE TORT, THOMAS DUBOS, Laboratoire Meteorologique Dynamique, FRANCOIS BOUCHUT, Laboratoire d'Analyse et de Mathematiques Appliquees, VLADIMIR ZEITLIN, Laboratoire Meteorologique Dynamique — Atmospheric and oceanic motion are usually modelled within the shallow-fluid approximation, which simplifies the 3D spherical geometry. For dynamical consistency, i.e. to ensure conservation laws for potential vorticity, energy and angular momentum, the horizontal component of the Coriolis force is neglected. Here new equation sets combining consistently a simplified shallow-fluid geometry with a complete Coriolis force are presented. The derivation invokes Hamilton's principle of least action with an approximate Lagrangian capturing the small increase with height of the solid-body entrainment velocity due to planetary rotation. A three-dimensional compressible model and a one-layer shallow-water model are obtained. The latter extends previous work done on the f-plane and β-plane. Preliminary numerical results confirm the accuracy of the 3D model within the range of parameters for which the equations are relevant. These new models could be useful to incorporate a full Coriolis force into existing numerical models and to disentangle the effects of the shallow-atmosphere approximation from those of the traditional approximation.

[1]This work was supported by the French ANR grant "SVEMO."

3:48PM L32.00002 On the Unexpected Longevity of the Great Red Spot, Oceanic Eddies, and other Baroclinic Vortices , PEDRAM HASSANZADEH, Harvard University, PHILIP MARCUS, UC Berkeley — Vortices in the ocean and atmosphere dissipate via various mechanisms such as wave emission, turbulence, and thermal radiation. However, these vortices are observed to live much longer than the time scales of the dissipation processes. Here we model these processes as either Rayleigh drag or Newtonian cooling with time scale τ, and use simulations of the 3D non-hydrostatic Boussinesq equations. Our results show that vortices in fact do NOT decay at the imposed time scale τ; they decay much slower, sometimes by a factor of 100. The slow decay is due to a meridional circulation, which converts the potential energy to the kinetic energy and vice versa and slows down the decay. In the presence of horizontal shear, the circulation can extract the shear energy and further energize the vortex. We explain the existence of the meridional circulation, the slow decay, and the resulting cyclone-anticyclone asymmetry using the numerical results, a physical model, and simplified equations. Our results suggest that the observed longevity of some vortices can be explained without a forcing mechanism. For very long-lived vortices, such as the Great Red Spot, our results imply that much weaker forcing, compared to what originally thought, is needed to maintain the vortices.

4:01PM L32.00003 The role of interactions between waves and baroclinic critical layers in zombie vortex self-replication , CHUNG-HSIANG JIANG, SUYANG PEI, UC Berkeley, PEDRAM HASSANZADEH, Harvard University, AARON WIENKERS, CALEB LEVY, PHILIP MARCUS, UC Berkeley — Inertio-gravity waves are triggered from various types of perturbations in numerical simulations of rotating, vertically-stratified and horizontal-shearing flows (Marcus et al. 2013 PRL). The interactions of these waves and baroclinic critical layers can create large vortices when the shear is sufficiently strong. An important feature of these flows is that an instability at one critical layer can excite an instability at its neighboring critical layers and spawn new generations of waves and vortices. Because the self-replication of these vortices in simulations of "dead zones" in protoplanetary disks reminds us of zombies multiplying by infecting each other, we call them "zombie vortices." However, not all interactions between waves and critical layers produce zombie vortices. The manner in which one "infected" critical layer infects its neighbor is not clear. The interaction of waves and critical layers are sensitive to the local Brunt-Vaisala frequency and to the wavelengths of the waves. Here we discuss how the interactions and formation of vortices depend upon the Brunt-Vaisala frequency (including its change in value as a function of vertical position) and our progress in understanding how the instability passes from a critical layer to its neighbor.

4:14PM L32.00004 On the Effects of Viscosity and Nonlinearity on Baroclinic Critical Layers , MENG WANG, CHUNG-HSIANG JIANG, UC Berkeley, PEDRAM HASSANZADEH, Harvard University, PHILIP MARCUS, UC Berkeley — A new family of baroclinic critical layers in rotating, stably-stratified flows plays a significant role in de-stabilizing shear flows and, in particular, is important in star formation in Keplerian disks (Marcus et al. PRL, 2013). These critical layers are characterized by singularities in their vertical velocities. To understand the critical layers, especially their thicknesses, and to help design future lab experiments that contain these layers, we use matched asymptotic expansions to obtain analytical solutions of the viscous flow in and around the baroclinic critical layers. To verify our solutions and to study the effect of nonlinearity, we also numerically simulate the critical layers produced by tilted vortices in strongly stratified fluids (no rotation, no imposed shear other than that induced by the vortex itself). This problem has been previously studied by Boulanger et al. (2008 JFM) and provides a framework to explore the physics of baroclinic critical layers before adding the complexity of rotation and strong shear.

4:27PM L32.00005 Evolution of a turbulence cloud under rotation[1] , AVISHEK RANJAN, P.A. DAVIDSON, Department of Engineering, University of Cambridge, UK — Localized regions of turbulence frequently occur in the geophysical environment and are governed by inhomogeneous dynamics. A direct numerical simulation study of such a region of turbulence (a "turbulence cloud") under external rotation is conducted at Rossby number of 0.1. The initial condition is generated using a spatial filter on a pre-run of fully developed homogeneous turbulence in a 512^3 periodic box. Lagrangian particle tracking is used to track turbulent diffusion. In the velocity iso-surfaces, columnar flow structures are seen to emerge from the turbulence cloud and grow linearly with time. These are created by inertial waves sustained by the Coriolis force in the rotating reference frame and propagate on a faster time scale compared to turbulence. Helicity is used as a diagnostic to confirm that columnar structures are indeed inertial waves.[2] The observations conform with the evolution of a single Gaussian eddy under rotation for which analytical solution is available in literature.

[1]The authors wish to thank P K Yeung for sharing his DNS code.

[2]Ranjan, A, Davidson, PA (2013) *Evolution of a turbulence cloud under rotation*, JFM (in preparation)

4:40PM L32.00006 Laboratory Scale Simulating of Spiral Plumes in the Mantle , ALBERT SHARIFULIN, Perm National Research Polytechnic University, ANATOLY POLUDNITSIN, Perm National Research University — On the basis of laboratory simulation a mechanism is established for the formation of the mantle convection spiral plumes from a core hot point in the presence of a roll-type large-scale convective flow. Experiment are close to fulfilling Golitsin's requirements [1] to laboratory models of mantle convection. We experimentally simulated the appearance of a plume from the local heat source generated by beam of green laser and study its interaction with cellular flow, simulating beneath the plates shear flow. It is shown that the presence of convective motion may lead to the formation of a strange spiral convective plume. Experimentally showed that the presence of cellular convective motion (simulating the large-scale shear flow exists beneath the plates) the plume from a point source of heat (core hot point) can acquire a spiral shape with horizontal sections needed to launch the mechanism of formation of chains of volcanic islands [2]. It is shown that possible existence of spiral plumes in mantle can help to interpret of last decade mantle tomography results.

[1] Golitsin GS (1979) Simple theoretical and experimental study of convection with some geophysical applications and analogies. J Fluid Mech 95: 567.
[2] Skilbeck, JN, Whitehead JA (1978) Formation of discrete islands in linear chains. *Nature* 272: 499.

4:53PM L32.00007 Direct numerical simulation of Coriolis effects on cylindrical gravity currents[1] , MARIANO CANTERO, Consejo Nacional de Investigaciones Cientificas y Tecnicas, JORGE SALINAS, Instituto Balseiro, UNCu-CNEA, THOMAS BONOMETTI, Institut de Mécanique des Fluides de Toulouse, ENZO DARI, Consejo Nacional de Investigaciones Cientificas y Tecnicas, Comision Nacional de Energia Atomica — Gravity currents are generated by the action of gravity (or other volumetric force) on changes in fluid density. When they appear in turbulent regime, gravity currents are of a non-linear nature and have a wide range of temporal and spatial scales. In these systems there is a strong coupling between turbulence and stratification effects, with important consequences in the exchange of mass, momentum and energy. At geophysical scale, the analysis of these type of flows is further complicated by the influence of rotation effects by the Coriolis forces originated by earth's rotation. In this work we address the rotational effects in gravity currents with cylindrical initial condition by means of direct numerical simulations (DNS). We report results on five three dimensional DNS with grid resolutions up to 166-million points, with different boundary conditions, Reynolds numbers (Re=4000 and Re=8000), and different conditions of rotation. The results focus mainly on the distance of propagation of the fronts, frequency of the successive outward fronts, and the turbulent structures present in the currents and their influence in flow dynamics.

[1]Support from CONICET, CNEA, ANPCyT, UPS and IMFT is greatly acknowledged

5:06PM L32.00008 Spontaneous bending of a columnar vortex in stratified-rotating fluids , EUNOK YIM, PAUL BILLANT, LadHyX, Ecole Polytechnique, CNRS — In a stably stratified and rotating fluids, an isolated asymmetric vortex can be unstable to a long-wavelength instability with an azimuthal wavenumber $m = 1$ which is different from classical instabilities. This instability bends the vortex and leads to the formation of pancake vortices. It can be the most dangerous instability when the background rotation and the stratification are strong. In order to better characterize this bending instability, numerical and asymptotic analyses have been performed for wide range of Froude and Rossby numbers and various velocity profiles. The maximum growth rate increases with the Rossby numbers but is independent of the Froude number when it is below unity. When the Froude number is above unity, the growth rate decays abruptly because of critical layers. By means of an asymptotic stability analysis for long-wavelength, we show that necessary instability conditions for any Froude and Rossby numbers are that there exists a critical radius r_c where the angular velocity is equal to the frequency $\Omega(r_c) = \omega$ and the vorticity gradient at the critical radius is positive $\zeta'(r_c) > 0$. These conditions are identical to those of the shear instability. Numerical results supporting these conditions will be presented.

5:19PM L32.00009 A new regime of instability for the stably stratified Taylor-Couette flow , PAUL BILLANT, JUNHO PARK, LadHyX, Ecole Polytechnique, CNRS — We show that the stably stratified Taylor-Couette flow is unstable when the angular velocity $\Omega(r)$ increases along the radial direction, a regime never explored before. The instability is different from the centrifugal instability: it is highly non-axisymmetric and involves the resonance of two families of inertia-gravity waves like for the strato-rotational instability. The growth rate is maximum when only the outer cylinder is rotating and goes to zero when $\Omega(r)$ is constant. The sufficient condition for linear, inviscid instability derived previously: $d\Omega^2/dr < 0$ is therefore extended to $d\Omega^2/dr \neq 0$, meaning that only the regime of solid-body rotation is stable in stratified fluids. A WKBJ analysis in the inviscid limit, confirmed by the numerical results, shows that the instability occurs only when the Froude number is below a critical value and only for a particular band of azimuthal wavenumbers. The physical mechanism of the instability will be explained in terms of wave over-reflection.

References: Park J. & Billant P., *J. Fluid Mech.*, **725**, 262-280 (2013); Yavneh, I., McWilliams, J. C. & Molemaker, M. J.,*J. Fluid Mech.* **448**, 1-21 (2001).

5:32PM L32.00010 Laboratory Observation of Stratorotational Instability with a Large Density Gradient , BRUCE RODENBORN, RUY IBANEZ, HARRY L. SWINNEY, Department of Physics and Center for Nonlinear Dynamics, University of Texas at Austin, Austin, Texas, USA — In 2001 a new class of instabilities in vertically stratified Taylor-Couette flows was predicted by Molemaker et al. (J. Fluid. Mech. **448**, 1). Dubrulle et al. (Astron. Astrophys. **429**, 1, 2005) then showed that this phenomenon, which they named stratorotational instability (SRI), could be a source of turbulence-producing angular momentum transport in an astrophysical accretion disk. Recently Shtemler et al. (Mon. Not. R. Astron. Soc. **406**, 517, 2010) showed that the SRI is unlikely to be a primary source of turbulence, but could well be an important secondary source. We use a Couette-Taylor system to study the SRI outside of the Boussinesq limit, i.e., with large axial density gradients, as exist in accretion disks. Our measurements of torque and the spatiotemporal structure of the flow as a function of the density profile and Froude number indicate that the SRI is robust outside of the Boussinesq limit, a minimum condition for relevance to accretion disks.

Monday, November 25, 2013 3:35PM - 5:45PM –

Session L33 Drops XI: Levitation and Propulsion on Surfaces 404 - Shreyas Mandre, Brown University

3:35PM L33.00001 Dynamic levitation of droplets , ANAIS GAUTHIER, CHRISTOPHE CLANET, DAVID QUERE, Physique et Mecanique des Milieux Heterogenes, CNRS, ESPCI, Paris France & Ladhyx, CNRS, Ecole Polytechnique, Palaiseau, France — We discuss how levitation can be induced for a liquid sitting on a plate in movement. In order to create the motion, we use a polished aluminum plate with a controlled rotational speed. As the surface reaches a critical velocity (between 1 and 10 m/s depending on the nature of the fluid), a drop gently deposited on the plate does not wet it but instead keeps a quasi-spherical shape, flying above the plate. We investigate experimentally the parameters that affect the value of the threshold speed between wetting and levitation, such as the nature of the fluid or the radius of the droplets. We also present a simple model to explain the existence of the levitation threshold.

3:48PM L33.00002 Levitation of a drop over a moving surface[1] , HENRI LHUISSIER, YOSHIYUKI TAGAWA, TUAN TRAN, CHAO SUN, POF - University of Twente - The Netherlands, POF TEAM — We investigate the levitation of a drop gently deposited onto the inner wall of a rotating hollow cylinder. For a sufficient velocity of the wall, the drop steadily levitates over a thin air film and reaches a stable angular position in the cylinder, where the drag and lift balance the weight of the drop. Interferometric measurements yield the three-dimensional air film thickness under the drop and reveal the asymmetry of the profile along the direction of the wall motion. A two-dimensional model is presented which explains the levitation mechanism, captures the main characteristics of the air film shape and predicts two asymptotic regimes for the film thickness h_0: For large drops $h_0 \sim \mathrm{Ca}^{2/3}\kappa_b^{-1}$, as in the Bretherton problem, where Ca is the capillary number based on the air viscosity and κ_b is the curvature at the bottom of the drop. For small drops $h_0 \sim \mathrm{Ca}^{4/5}(a\kappa_b)^{4/5}\kappa_b^{-1}$, where a is the capillary length.

[1]We thank Detlef Lohse for the opportunity to carry this work.

4:01PM L33.00003 Bouncing and rolling motions of capillary Leidenfrost drops on a micro-ratchet , KYRA STEPHANOFF, PAUL STEEN, Cornell University, HENRI LHUISSIER, DETLEF LOHSE, Physics of Fluids, University of Twente — Capillary drops falling onto a micro-ratchet that is heated to temperatures 275 C $\leq$ T $\leq$ 350 C, bounce off a layer above the ratchet multiple times before settling down to the motion typically observed when the Leidenfrost effect is present. The deformation of the drops is asymmetrical about the y-z plane through the center of the drops as the drops move in the x direction. The magnitude of the asymmetrical deformations varies with ratchet temperature as does the number of bounces. Videos show that, when a drop settles down to its "rolling" regime, the fluid within a drop moves in a counter-clockwise direction. The counter-clockwise internal motion and the asymmetric deformations of a bouncing drop indicate, albeit indirectly, that the fluid in the ratchet cavities is moving clockwise. A simple model that correlates well with the experimental observations is presented.

4:14PM L33.00004 Propulsion on a superhydrophobic ratchet , PHILIPPE BOURRIANNE, GUILLAUME DUPEUX, CHRISTOPHE CLANET, DAVID QUERE, PMMH, ESPCI / LadHyX, Ecole Polytechnique — As shown by Linke in 2006, an evaporating Leidenfrost drop self-propels on a hot ratchet. Indeed, the vapour flow below the drop can be rectified by the asymmetric teeth of the ratchet and, therefore, entrain the levitating drop by viscosity. This motion is usually observed above the Leidenfrost temperature. We show how the use of a super-hydrophobic ratchet allows us to extend self-propulsion down to the boiling point of water, and even below. We discuss a possible explanation for this "cold regime" of propulsion.

4:27PM L33.00005 Surfing on a herringbone , DAN SOTO, GUILLAUME LAGUBEAU, CHRISTOPHE CLANET, DAVID QUERE, PMMH/Ladhyx — Liquids in the Leidenfrost state levitate on hot solids, owing to the formation of a cushion of vapor. Without contact, drops glide with negligible friction on their substrate. The conjunction of vapor production and frictionless motion can be exploited to self-propel liquids when placed on hot horizontal ratchets. It was proposed to understand the effect as follows: the asymmetric teeth of the ratchet rectify the vapor flow below the levitating liquid, which is entrained by the viscous vapor. In our presentation, we propose to induce similar effects by geometrical means, hence achieving new designs for self-propelling Leidenfrost liquids. We force a directional flow of vapor by etching a herringbone pattern in the hot substrate. We show how this design can be tuned to optimize the propelling force and the drop speed, which is quantitatively analyzed. We eventually extend these principles to self-propel plastic levitating cards at room temperature, using patterned hockey tables.

4:40PM L33.00006 The Walking Droplet Instability , JOSHUA BOSTWICK, Northwestern University, PAUL STEEN, Cornell University — A droplet of liquid that partially wets a solid substrate assumes a spherical-cap equilibrium shape. We show that the spherical-cap with a mobile contact-line is unstable to a non-axisymmetric disturbance and we characterize the instability mechanism, as it depends upon the wetting properties of the substrate. We then solve the hydrodynamic problem for inviscid motions showing that the flow associated with the instability correlates with horizontal motion of the droplet's center-of-mass. We calculate the resulting "walking speed." A novel feature is that the energy conversion mechanism is not unique, so long as the contact-line is mobilized. Hence, the walking droplet instability is potentially significant to a number of industrial applications, such as self-cleansing surfaces or energy harvesting devices.

4:53PM L33.00007 Propelling a water drop with the vapor-mediated Marangoni effect[1] , SEUNGHO KIM, HO-YOUNG KIM, Seoul National University — We show that a water drop on solid surfaces can be propelled just by placing a volatile alcohol drop nearby. It is found to be because the water-air interface near the alcohol drop mixes with alcohol vapor, thereby locally lowering the surface tension. The surface-tension-gradient induces the motion of the water drop, enabling the trajectory control of water drops through the motion of remote alcohol drops. This vapor-mediated Marangoni effect also gives rise to other interesting interfacial flow phenomena, such as nucleation of holes on a water film and ballooning of a water drop hanging from a syringe needle with the approach of an alcohol drop. We visualize such interfacial dynamics with a high-speed camera and rationalize their salient features by scaling analysis.

[1]This work was supported by the National Research Foundation of Korea (grant no. 2012-008023).

5:06PM L33.00008 Thermocapillary-driven motion of a droplet on an inclined substrate: contact line dynamics, and non-monotonic dependence of surface tension on temperature , GEORGE KARAPETSAS, University of Thessaly, KIRTI SAHU, Indian Institute of Technology, Hyderabad, KHELLIL SEFIANE, University of Edinburgh, OMAR MATAR, Imperial College London — We consider the two-dimensional motion of a droplet on an inclined, non-isothermal solid substrate. We use the lubrication approximation to obtain a single evolution equation for the interface, which accounts for gravity, capillarity, and thermo-capillarity, brought about by the dependence of the surface tension on temperature. For the latter, a nonlinear function is used, which exhibits a well-defined minimum. The contact line motion is modelled by coupling the contact line speed to the difference between the dynamic and equilibrium contact angles; the latter vary dynamically during the droplet motion through the dependence of the liquid-gas, liquid-solid, and solid-gas surface tensions on the local contact line temperature. Thus, the local substrate wettability also varies dynamically at the two edges of the drop. A full parametric study is carried out for constant substrate temperature gradients in order to investigate the interplay between Marangoni stresses, induced by thermo-capillarity, gravity, and contact line dynamics in the presence of local wettability variations, and non-monotonic dependence of the surface tension on temperature. The results of this study are presented together with comparisons against experimental data.

5:19PM L33.00009 Pancake droplets on the grill: Thermocapillary motion of confined droplets in Hele-Shaw cells[1] , MARC HABISREUTINGER, FRANÇOIS GALLAIRE, PIERRE-THOMAS BRUN, MATHIAS NAGEL, EPFL - STI - IGM - LFMI — Assuming constant surface tension, the sphere is an equilibrium shape for a drop that is translating at low Reynolds number through an infinite domain of stationary fluid. Remarkably, this result still holds true for the thermocapillary motion of a drop in a fluid at rest, when a constant temperature gradient is imposed and convection effects are neglected (Young, Goldstein and Block [1]). On the contrary, we show analytically that such an ideal result no longer stands when constraining the surrounding geometry of the droplet. For instance, flattened cylindrical droplets in microchannels, so-called pancake droplets, do not represent an equilibrium shape of thermocapillary motion. Our numerical studies also enable to take into account the flow-induced temperature variations and could help building a scaffold towards the individual control of droplets in microchannels.

[1] N.O. Young, J.S. Goldstein and M.J. Block, The motion of bubbles in a vertical temperature gradient, J. Fluid Mech. 6, 350-6 (1959).

[1]This project is partially funded by the European Research Council.

5:32PM L33.00010 Thermocapillary flow induced by the optical irradiation of carbon nanoparticles , J. RODRIGO VELEZ CORDERO, JUAN A. HERNANDEZ CORDERO, Universidad Nacional Autonoma de Mexico — Transport of discrete drops in micro-channels constitutes an essential operation in microfluidic devices. Due to the small characteristic size of micro-channels, the pressure drop necessary to induce motion can be produced by surface tension forces, according to Laplace law, and not only by the use of a mechanical pump. Thermocapillary motion is produced when one extreme of the drop is heated: since surface tension diminishes with temperature, the pressure difference on both extremes will be unbalanced and subsequent equilibrated by the motion of the drop. In this work we used thermocapillary pumping to induce the motion of drops by using a polymeric matrix embedded with carbon nanoparticles (PDMS-Cpart) capable to absorb radiative energy (delivered by an optical fiber) and operate as a heat source. Capillaries with different sizes were then coated with the PDMS-Cpart mixture. The observed motion of the drops, whose velocity is comparable to those achieved using metallic heaters, was analyzed under three important considerations: the dynamic angle hysteresis, optical depth of the PDMS-Cpart layer, and the optical power delivered by the optical fiber.

Monday, November 25, 2013 3:35PM - 5:45PM –

Session L34 Drops XII: Elastic Surfaces and Fibers 405 - Siddhartha Das, University of Alberta

3:35PM L34.00001 Dew-driven folding of insect wings , ANDREW DICKERSON, SAM BEADLES, COURTNEY CLEMENT, DAVID HU, Georgia Institute of Technology — Small insect wings fold into tacos when exposed to dewfall or fog for extended times. Such shapes are tightly held together and require great force or long evaporation times for the wings to unfold. In this experimental investigation, we use time-lapse and high-speed videography on a mosquito wing exposed to fog to characterize the folding process from a flat wing to a taco. We observe a taco is formed through a series of processes involving wing bending, unbending, and subsequent tight folding of the wing following the sliding of the drop off the wing. We use a simplified 2D model to determine the forces coalescing drops exert on the wing, and present folding-resistant design suggestions for micro-aerial vehicle wings.

3:48PM L34.00002 Aperture-Embedded Polymer Microlenses for Ultra-Low-Cost Microscopy Platforms (Foldscope)[1] , LAUREL KROO, GEORGE K. HERRING, MANU PRAKASH, Stanford University — Our lab has recently introduced an ultra-low cost microscopy platform: Foldscope, an origami based print-and-fold paper microscope for applications in disease diagnostics and science education. This current study introduces the concept of aperture-embedded microlenses made of ultra-violet curable polymers as a solution for high-throughput roll-to-roll manufacturing of micro-optical components utilized in Foldscope. The approach employs fluid droplets trapped in an aperture via capillary forces to induce various characteristic lens surfaces. By implementing static and dynamic pressure as a method to manipulate the droplet, a large and versatile range of optical surface shapes become viable. When the polymer droplet acquires the desired shape on either side of the aperture, the lens is frozen *in situ* within milliseconds with a high-power UV source. We explore the dynamics of ultra-fast curing of polymeric droplets through both experimental and analytical means. The presented capillary induced printed lens manufacturing enables ultra-low cost optical instruments.

[1]Prakash Labs, Stanford University

4:01PM L34.00003 Interaction of Drops on a Soft Substrate , LUUK A. LUBBERS, JOOST H. WEIJS, Physics of Fluids Group, University of Twente, P.O. Box 217, 7500 AE Enschede, The Netherlands, SIDDHARTHA DAS, Department of Mechanical Engineering, University of Alberta, Alberta, Canada T6G 2G8, LORENZO BOTTO, Department of Chemical Engineering, Imperial College, London, United Kingdom, BRUNO ANDREOTTI, Physique et Mecanique des Milieux Hétérogènes, UMR 7636 ESPCI -CNRS, Univ. Paris-Diderot, 10 rue Vauquelin, 75005, Paris, France, JACCO H. SNOEIJER, Physics of Fluids Group, University of Twente, P.O. Box 217, 7500 AE Enschede, The Netherlands — A sessile drop can elastically deform a substrate by the action of capillary forces. The typical size of the deformation is given by the ratio of surface tension and the elastic modulus, γ/E, which can reach up to 10-100 microns for soft elastomers. In this talk we theoretically show that the contact angles of drops on such a surface exhibit two transitions when increasing γ/E: (i) the microsocopic geometry of the contact line first develops a Neumann-like cusp when γ/E is of the order of few nanometers, (ii) the macroscopic angle of the drop is altered only when γ/E reaches the size of the drop. Using the same framework we then show that two neighboring drops exhibit an effective interaction, mediated by the deformation of the elastic medium. This is in analogy to the well-known Cheerios effect, where small particles at a liquid interface attract eachother due to the meniscus deformations. Here we reveal the nature of drop-drop interactions on a soft substrate by combining numerical and analytical calculations.

4:14PM L34.00004 Elastically dominated viscous spreading , JEROME NEUFELD, BP Institute, Department of Earth Sciences, Department of Applied Mathematics and Theoretical Physics, University of Cambridge, MARCIN MALINOWSKI, Department of Physics, University of Cambridge — The spreading of viscous liquid drops on a floating elastic sheet is a rich problem exhibiting striking new phenomena across a wide array of scales, from the buckling of nanoscale elastic sheets to the deformation of the Indian subcontinent by the Tibetan plateau. Here we show that when density of the fluid exceeds that of the "ocean" supporting the elastic sheet ($\rho_f > \rho_o$) a steady state radius is reached. In contrast, for relatively light liquid droplets ($\rho_f < \rho_o$) two modes of propagation are found. At early times bending provides a small correction to the classic viscous gravity current, while at late times a new bending-dominated mode of propagation emerges. The results are applicable to the spreading of droplets on elastic membranes and, at very much larger scales, to the dynamics and topology of the Tibetan plateau.

4:27PM L34.00005 Droplet impact on an elastic beam; a leaf-raindrop system , SEAN GART, DANIEL CHIQUE, SUNGHWAN JUNG, Virginia Tech, BIOLOGICALLY INSPIRED FLUIDS LAB TEAM — We investigate a leaf-drop system exhibiting a unique system of coupled elasticity and drop dynamics by studying water droplet impact on an elastic cantilever beam with a wettable and non-wettable surface. We found that wettable beams experience much higher torque and bending energy than non-wettable beams. This is because a drop sticks to a wettable beam and rolls off of a non-wettable beam. Simple analytical models can explain the difference in bending energy and torque of wettable and non-wettable beams, which is verified with experimental observations.

4:40PM L34.00006 Compound droplets on fibers , FLORIANE WEYER, LAURENT DREESEN, MARJORIE LISMONT, NICOLAS VANDEWALLE, GRASP, University of Liege, B4000 Liege, Belgium — The development of a fiber-based digital microfluidics device mimicking biological membranes requires the formation and the manipulation of compound droplets in order to prevent contamination and evaporation. In the present work, we propose a study of compound water-oil droplets on fibers. Although the case of pure droplets is well established, we show herein that the situation becomes more complex for compound fluidic systems. In particular conditions, contact lines are merging and quadruple contact lines are formed. Depending on the formation of this quadruple line, the behavior of the compound droplet is different from pure systems. Two different aspects are then addressed: the equilibrium position and the maximum size of the droplets on the fiber, being obtained by the balance of capillary and gravity forces. Finally, we show that the characteristic length is the fiber diameter or the size of the core droplet depending on whether a quadruple line is formed.

4:53PM L34.00007 Wetting and drying of liquid on crossed fibers , ALBAN SAURET, ALISON D. BICK, HOWARD A. STONE, Princeton University, COMPLEX FLUIDS GROUP TEAM — Fibrous media are common in various engineered systems such as filters, paper or the textile industry. Many of these materials can be described as a network of fibers in which a wetting liquid tends to accumulate at its nodes and changes the bulk properties. Here we study a drop of silicone oil sitting on the simplest element of the array: two rigid crossed fibers. In particular, we investigate experimentally how the structure of the material affects the wetting and drying dynamics of that liquid drop. We first show that the liquid can adopt different shapes from a long liquid column to a drop. The transition between these morphologies depends on the volume of liquid, the tilting angle between the fibers, as well as the fiber radius. The wetting length in the column state can be predicted analytically. Because of these different shapes, the liquid exhibits different drying kinetics, which effects the overall drying time. Our study suggests that shearing a wetted array of fibers, by tuning the liquid morphology, may enhance the drying rate.

5:06PM L34.00008 Drops moving along and across a filament[1] , RAKESH P. SAHU, SUMAN SINHA-RAY, ALEXANDER YARIN, University of Illinois at Chicago, BEHNAM POURDEYHIMI, North Carolina State University — The present work is devoted to the experimental study of oil drop motion both along and across a filament due to the air jet blowing. In case of drop moving along the filament, phenomena such as drop stick–slip motion, shape oscillations, shedding of a tail along the filament, the tail capillary instability and drop recoil motion were observed which were rationalized in the framework of simplified models. Experiments with cross-flow of the surrounding gas relative to the filament with an oil drop on it were conducted, with air velocity in the range of 7.23 to 22.7 m s^{-1}. The Weber number varied from 2 to 40 and the Ohnesorge number varied from 0.07 to 0.8. The lower and upper critical Weber numbers were introduced to distinguish between the beginning of the drop blowing off the filament and the onset of the bag-stamen breakup. The range of the Weber number between these two critical values is filled with three types of vibrational breakup: V1 (a balloon-like drop being blown off), V2 (a drop on a single stamen being blown off), and V3 (a drop on a double stamen being blown off). The Weber number/Ohnesorge number plane was delineated into domains of different breakup regimes.

[1]The work is supported by the Nonwovens Cooperative Research Center (NCRC).

5:19PM L34.00009 Drops in wedges , ETIENNE REYSSAT, PMMH, ESPCI — Constrictions or widening of porosity result in non-balanced capillary forces acting at the interface between two fluid phases in porous media. Gradients of confinement can thus be used to produce, break up or manipulate drops and bubbles. We investigate experimentally the motion of oil drops and air bubbles confined between two quasi-horizontal plates forming a sharp wedge. The confinement gradient drives the migration of drops of wetting fluid toward the apex of the wedge. The capillary driving force is balanced by viscous dissipation occurring both in the bulk of the drop and along the contact lines. We provide a minimal model that quantitatively explains the migrations dynamics. In particular, we observe and explain two asymptotic regimes associated to both dissipation modes. We also present various possibilities to trap, expel or transport fluid using confinement gradients.

5:32PM L34.00010 Snail droplets: How fast is a flattened droplet transported by a more viscous wetting carrier fluid in a thin microchannel? , FRANCOIS GALLAIRE, MATHIAS NAGEL, LFMI EPFL Lausanne — It has been known for more than hundred years that a spherical droplet of fluid #1 immersed in an unconfined environment of a more viscous carrier fluid #2 travels at a relative velocity outreaching the carrier fluid. This result does not hold when the droplet is squeezed in-between walls, an ubiquitous situation in microfluidics. Indeed, the presence of confining walls results in thin lubricating films of fluid #2 lying in-between the walls and the droplet interface, which introduce an additional source of drag that increases as the film thickness decreases. Following Park and Homsy (1984), the lubrication film thickness may be shown to vary as $Ca^{2/3}$ where Ca is the capillary number that compares the viscous damping and surface tension effects. These films also affect the pressure jump across the interface, which, combined with the Brinkman equations for the flow in Hele-Shaw cells, allows to determine the flow field and the resulting deformations of the droplet interface. The obtained results appear to be in good agreement with experimental measurements. A multipole expansion of the flow field created by the droplet is then coupled to the nonlinear boundary condition at the interface yielding a simple analytical expression for the relative droplet velocity.

Monday, November 25, 2013 3:35PM - 5:45PM –

Session L35 Chaos, Fractals and Dynamical Systems III: Miscellaneous Topology and Model Characterization 406 - Steve Brunton, University of Washington

3:35PM L35.00001 Describing Chaotic Dynamics in Experimental Rayleigh-Bénard Convection Using Persistent Homology Theory[1] , JEFFREY TITHOF, BALACHANDRA SURI, Georgia Institute of Technology, MIROSLAV KRAMAR, VIDIT NANDA, Rutgers University, MU XU, MARK PAUL, Virginia Tech, KONSTANTIN MISCHAIKOW, Rutgers University, MICHAEL SCHATZ, Georgia Institute of Technology — We employ a new technique for describing the dynamics of spatiotemporal chaos in Rayleigh-Bénard convection. We collect shadowgraph images of multiple time series of weakly chaotic flows, each starting from similar initial conditions which we impose using a laser. We then encode the topological characteristics of each frame into a so-called persistence diagram, measure the distance across all diagrams, and study the dynamical behavior. Results are compared to similar analyses of simulation data. This new methodology provides unique insight into the time evolution of this dynamical system and the chaotic evolution across separate runs, in both experiment and simulation.

[1]This work is supported under NSF grant DMS-1125302.

3:48PM L35.00002 Analyzing the dynamics of pattern formation in the space of persistence diagrams , MIROSLAV KRAMAR, KONSTANTIN MISCHAIKOW, Rutgers, MICHAEL SCHATZ, JEFFREY TITHOF, Georga Tech, MARK PAUL, MU XU, Virginia Tech — Persistence diagrams are a relatively new topological tool for describing and quantifying complicated patterns in a simple but meaningful way. We will demonstrate this technique on patterns appearing inRayleigh-Benard convection. This procedure allows us to transform experimental or numerical data from experiment or simulation into a point cloud in the space of persistence diagrams. There are a variety of metrics that can be imposed on the space of persistence diagrams. By choosing different metrics one can interrogate the pattern locally or globally, which provides deeper insight into the dynamics of the process of pattern formation. Because the quantification is being done in the space of persistence diagrams this technique allows us to compare directly numerical simulations with experimental data.

4:01PM L35.00003 Burning Invariant Manifold Theory and the Bipartite Digraph Representation of Generalized Dynamical System Formed by One-way Barriers[1]

, JOHN LI[2], University of Southern California, JOHN MAHONEY, KEVIN MITCHELL, University of California, Merced, TOM SOLOMON COLLABORATION[3] — The recently developed *Burning Invariant Manifold* (BIM) theory took a dynamical system approach to understand front propagation in *Advection-Reaction-Diffusion* systems and successfully predicted both the short-term and asymptotic front behavior by finding the unstable BIMs which act as barriers to front propagation. Unlike separatrices in traditional dynamical system being two-way barriers, the BIMs are one-way barriers. This asymmetry gives rise to a much richer dynamical behavior than traditional dynamical systems. Through numerical simulations, we found that the stable BIMs are the basin boundaries. Based on the properties of BIM theory, we further derived a theory to investigate a dynamical system consists of one-way barriers and the cooperative behavior of these barriers. This theory reveals the global structure of both stable and unstable BIMs by first using a systematic algorithm to convert the flow to a bipartite digraph and then extracting information of the steady states of fronts and corresponding basins of attraction from the digraph.

[1]This work was supported by the US National Science Foundation under grant PHY-0748828 and NSF Fellowship DGE-0937362.

[2]Previous Affiliation: University of California, Merced

[3]tsolomon@bucknell.edu Department of Physics & Astronomy Bucknell University

4:14PM L35.00004 Experimental studies of mixing barriers and reaction fronts in a steady, three-dimensional flow[1]

, HARRISON MILLS, TOM SOLOMON, Bucknell University — We present experiments studying chaotic mixing and front propagation in a steady, three-dimensional (3D) flow composed of nested vortices. Passive mixing is characterized by tracking almost-neutral, fluorescent tracer particles in the flow. A fluorescent dye is also used, and the spreading of this dye is monitored with a scanning laser system and a camera that images a stack of cross-sectional images. Using both methods, we find evidence of both ordered and chaotic regions of mixing in the flow. We also present preliminary results of studies of behavior of the Ruthenium-catalyzed, excitable Belousov-Zhabotinsky chemical reaction in this flow. Propagating fronts of this reaction are characterized in 3D by the same laser-scanning system. The goal of these experiments is to determine barriers to front propagation and to compare these reaction barriers to the barriers observed for passive mixing in the same flow. Ultimately, a generalization of the *burning invariant manifold* theory[2] to 3D will be used to explain these barriers.

[1]Supported by NSF Grants DMR-1004744 and PHY-1156964.

[2]J. Mahoney, D. Bargteil, M. Kingsbury, K. Mitchell and T. Solomon, Europhys. Lett. **98**, 44005 (2012).

4:27PM L35.00005 Pinning of reaction fronts by burning invariant manifolds[1]

, PETER MEGSON[2], TOM SOLOMON, Bucknell University — We present experiments that study the behavior of the excitable Belousov-Zhabotinsky chemical reaction in a translating, regular array of vortices. In a reference frame moving with the translating vortices, the flow is equivalent to a stationary vortex array with an imposed uniform wind. Under a wide range of wind speeds, reaction fronts pin to the vortex flow, neither propagating forward against the wind nor being blown back. We explain this pinning behavior with the use of a recent theory[3] of *burning invariant manifolds* (BIMs) that act as one-way barriers against any propagating reaction front. When the reaction fronts are pinned, several BIMs combine to form an extended barrier that determines the shape of the pinned fronts. The location of the BIMs are calculated numerically with an analytical approximation of the velocity field and are compared with experimental images of the pinned fronts. We also study transient behavior that helps elucidate the one-way nature of the BIMs.

[1]Supported by NSF Grants DMR-1004744 and PHY-1156964.

[2]Current address: Harvey Mudd College, Claremont, CA

[3]J. Mahoney, D. Bargteil, M. Kingsbury, K. Mitchell and T. Solomon, Europhys. Lett. **98**, 44005 (2012).

4:40PM L35.00006 Deterministic Aperiodic Sickle Cell Blood Flows

, LOUIS ATSAVES[1], WESLEY HARRIS[2], MIT — In this paper sickle cell blood flow in the capillaries is modeled as a hydrodynamical system. The hydrodynamical system consists of the axisymmetric unsteady, incompressible Navier-Stokes equations and a set of constitutive equations for oxygen transport. Blood cell deformation is not considered in this paper. The hydrodynamical system is reduced to a system of non-linear partial differential equations that are then transformed into a system of three autonomous non-linear ordinary differential equations and a set of algebraic equations. We examine the hydrodynamical system to discern stable/unstable, periodic/nonperiodic, reversible/irreversible properties of the system. The properties of the solutions are driven in large part by the coefficients of the governing system of equations. These coefficients depend on the physiological properties of the sickle cell blood. The chaotic nature of the onset of crisis in sickle cell patients is identified.

[1]Research Assistant

[2]Professor of Aeronautics and Astronautics

4:53PM L35.00007 Bifurcation analysis of an oscillating cylinder wake[1]

, MATTHEW CHU CHEONG, JONATHAN TU, CLARENCE ROWLEY, Princeton University — The flow past a transversely oscillating cylinder gives rise to distinct vortex patterns in the wake, with the particular pattern depending on Reynolds number, Strouhal number, and reduced frequency. In this work, we perform a numerical bifurcation analysis of the transitions between 2S, P+S, and disordered wakes at Reynolds numbers 100. Due to the high dimensionality of fluid flow simulations, standard tools such as AUTO are not applicable. Instead, we turn to Krylov-subspace-based algorithms. The coherent wake patterns (2S, P+S) are stable periodic orbits whose common period is that of the forced oscillation. To identify bifurcations, we perform stability analyses of the Poincare map by stroboscopically sampling the flow. We find the following bifurcations as the reduced frequency is held constant and the Strouhal number is increased : (1) the transition from a 2S wake to a P+S wake is a supercritical pitchfork bifurcation, (2) the transition from a P+S wake back to a 2S wake is another supercritical pitchfork bifurcation, and (3) the transition from a 2S wake to a disorganized wake is a torus bifurcation. Consistent with these bifurcations, we confirm the existence of unstable 2S wakes at Strouhal numbers where P+S and disordered wakes are observed.

[1]This work was supported by the AFOSR and the National Science Foundation.

5:06PM L35.00008 Probing the dynamics of Rayleigh-Bénard convection using numerical simulations for the conditions of experiment

, MU XU, Virginia Tech, JEFFREY TITHOF, Georgia Institute of Technology, MIRO KRAMAR, Rutgers University, BALACHANDRA SURI, Georgia Institute of Technology, VIDIT NANDA, Rutgers University, MICHAEL SCHATZ, Georgia Institute of Technology, KONSTANTIN MISCHAIKOW, Rutgers University, MARK PAUL, Virginia Tech — We present results from large-scale parallel numerical simulations of Rayleigh-Bénard convection for the precise conditions of experiment. We are interested in cylindrical convection domains of moderate aspect ratio with a Prandtl number of order 1. We compute the leading order Lyapunov vector and Lyapunov exponent and use these to quantify the dynamics. We explore time periodic dynamics and also the breakdown of patterns with prescribed initial conditions toward weakly chaotic dynamics. We directly compare our results from numerical simulation with experimental measurements where possible. The numerics yield physical insights into the spatiotemporal dynamics of convection that we can use to connect with ideas from topology such as persistence diagrams.

5:19PM L35.00009 Chaotic flow and the finite-time Lyapunov exponent: Competitive autocatalytic reactions in advection-reaction-diffusion systems , RICHARD M. LUEPTOW, CONOR P. SCHLICK, PAUL B. UMBANHOWAR, JULIO M. OTTINO, Northwestern University — We investigate chaotic advection and diffusion in competitive autocatalytic reactions. To study this subject, we use a computationally efficient method for solving advection-reaction-diffusion equations for periodic flows using a mapping method with operator splitting. In competitive autocatalytic reactions, there are two species, B and C, which both react autocatalytically with species A (A+B→2B and A+C→2C). If there is initially a small amount of spatially localized B and C and a large amount of A, all three species will be advected by the velocity field, diffuse, and react until A is completely consumed and only B and C remain. We find that the small scale interactions associated with the chaotic velocity field, specifically the local finite-time Lyapunov exponents (FTLEs), can accurately predict the final average concentrations of B and C after the reaction is complete. The species, B or C, that starts in the region with the larger FTLE has, with high probability, the larger average concentration at the end of the reaction. If species B and C start in regions having similar FTLEs, their average concentrations at the end of the reaction will also be similar. Funded by NSF Grant CMMI-1000469.

5:32PM L35.00010 On a novel approach to anomalous transport in turbulent fluid and plasma , DHURJATI PRASAD DATTA, Department of Mathematics, University of North Bengal — New nonclassical self similar intermediate asymptotics considered recently [1,2] in the context of linear differential equations are shown to have interesting applications in offering a novel explanation of the origin of anomalous transport phenomena in turbulent flows in fluids and plasma devices. The intermediate asymptotics, in the late time or in the inviscid limit, conspire to produce smooth multifractal measures on a turbulent fluid medium leading naturally to generation of stretched Gaussian distributions for passive scalar tracer concentration from the turbulent, integral order, advection-diffusion equation. Such heavy tailed stretched Gaussian distributions can explain the observed anomalous scaling of the average and mean square displacements of tracer particles in a turbulent medium.We also point out that the present novel mechanism for generation of multifractal measure can actually be interpreted as a new class of instabilities leading to turbulence.

[1] D. P. Datta, On a novel signature of late time asymptotics: a new route to nonlinearity, (2013), Communicated.
[2] D. P. Datta, Novel Late Time Asymptotics: Applications to Anomalous Transport in Turbulent Flows, REDS,(2013), accepted.

Monday, November 25, 2013 3:35PM - 5:45PM –

Session L36 Flow Control VI: Systems and Mechanisms 407 - John Kuhlman, West Virginia University

3:35PM L36.00001 An active flow control theory of the vortex breakdown process , ZVI RUSAK, JOSHUA GRANAT, Rensselaer Polytechnic Institute, SHIXIAO WANG, U. Auckland NZ — An active flow control theory of the vortex breakdown process in incompressible swirling flows in a finite-length, straight, circular pipe is developed. Flow injection distributed along the pipe wall is used as the controller. The flow is subjected to non-periodic inlet-outlet conditions. A long-wave asymptotic analysis results in a nonlinear model problem for the dynamics and control of both inviscid and high Reynolds number flows. The approach provides the bifurcation diagram of steady states and the stability characteristics of these states. In addition, an energy analysis of the controlled flow dynamics suggests a feedback control law which relates the flow injection to the evolving maximum radial velocity at the inlet. The feedback control cuts the natural feed forward mechanism of the breakdown process. Computed examples based on the full Euler and NS formulations at various swirl levels demonstrate the evolution to near-steady breakdown states when swirl is above a critical level. Moreover, applying the proposed feedback control law during flow evolution, shows for the first time the successful elimination of the breakdown states and flow stabilization on an almost columnar state for a wide range of swirl, up to 30 percent above critical.

3:48PM L36.00002 Flow control using ferrofluids , FRANCOIS CORNAT, DAVID BECK, IAN JACOBI, HOWARD STONE, Princeton University — A novel flow control technique is proposed which employs a ferrofluidic lubricant infused in a micro-patterned substrate as a "morphing surface" for control of wall-bounded flows. Traditionally, morphing surfaces produce dynamic changes in the curvature and roughness of solid substrates for active control of high Reynolds number flow features such as boundary layer separation and turbulent streaks. We show how these surface modifications can be achieved with a thin liquid layer in the presence of a normal magnetic field. By impregnating a chemically-treated, micro-patterned surface with a fluorinated ferrofluid, the fluid is maintained as a thin super-hydrophobic film and can be redistributed on the substrate by magnetic forces to dynamically reveal or conceal the underlying surface roughness. Moreover, the surface topography of the ferrofluid film itself can be modified to produce an enhanced roughness, beyond the scale of the underlying substrate pattern. Both types of ferrofluidic surface modifications are studied in micro- and macro- scale channels in order to assess the feasibility of flow modification at low to moderate Reynolds numbers.

4:01PM L36.00003 Vibration Amplitude of a Flexible Filament Changes Non-Monotonically with Angle of Attack , H. DOGUS AKAYDIN, Stanford University, CEES J. VOESENEK, Wageningen University, DAVID LENTINK, Stanford University, LENTINK LAB TEAM — Certain animals exploit the interaction of their flexible, foil-like appendages with vortices to propel themselves effectively in surrounding fluids. The interaction is reciprocal because the fluid forces deform the appendage while the appendage alters the flow. As the flow separates from a foil-like structure, it rolls-up into a vortex with a low-pressure core, which deforms the structure until the vortex is shed into the wake. A control over this interaction can provide a robust aerodynamic performance. To identify the salient parameters that control fluid-structure interactions on a deformable structure, we varied the thickness, length and angle of attack of a rubber filament in a quasi-two-dimensional flow generated using a soap-film tunnel. We resolved fluid-structure interaction by simultaneously measuring deformation of the filament and motion of particles in the fluid using high-speed cameras. We show that increasing length or decreasing diameter of the filament increases its vibration amplitude monotonically. In stark contrast, the angle of attack of the filament may alter the amplitude of vibrations in a non-monotonic way: Within a certain range of angle of attack, the filament motion and vortex shedding lock-in, i.e. become synchronized, and result in a violent flapping behavior. This response can therefore be ceased not only by decreasing but also by increasing the angle of attack at the leading edge. Such an insight can help us engineer more effective bio-inspired robots, energy harvesters, and flow control devices with vibration control.

4:14PM L36.00004 Control of a Separation bubble at Low Reynolds Numbers Using Electro-Active Polymers , HALEY DELL'ORSO, LUCIA CHANG, SARAH ZAREMSKI, EDWARD DEMAURO, CHIA LEONG, MICHAEL AMITAY, Rensselaer Polytechnic Institute — An experimental investigation was performed to study the effects of electro-active polymers (EAPs) on a 3-dimensional separation bubble on a two-dimensional NACA0009 airfoil at a Reynolds number of 20,000 and an angle of attack 5 deg. A single row of EAPs was placed at 20% chord and activated at 1500V and 50Hz, corresponding to the Kelvin-Helmholtz frequency of the separated mixing layer. Stereoscopic Particle Image Velocimetry data were collected in the vicinity of the EAPs for three cases: baseline (no EAP present), EAP present but not actuated, and EAP present and actuated. Data demonstrated that the presence of the EAP slightly reduced the magnitude of the separation bubble. When the EAPs were actuated at the chosen frequency and voltage, the separation bubble was almost completely mitigated.

4:27PM L36.00005 Effect of Boundary Layer Thickness on Secondary Structures in a Short Inlet Curved Duct , JEREMY GARTNER, MICHAEL AMITAY, Rensselaer Polytechnic Institute — The flow pattern in short ducts with aggressive curvature can lead in some cases to an asymmetric flow field. In the current work, a two dimensional honeycomb mesh was added upstream of the curved duct to create a pressure drop across it, and therefore an increased velocity deficit in the boundary layer profile. This velocity deficit led to a stronger streamwise separation, overcoming the flow mechanisms that result in the asymmetric flowfield. Experiments were conducted at $\mathbf{M} = 0.2$, 0.44 and 0.58 in an expanding aggressive duct with square cross section with an area ratio of 1.27. Pressure data, together with Particle Image Velocimetry (PIV), verify the symmetry of the incoming flow field. Steady pressure distributions along the lower surface of the curved duct were obtained, as well as steady and time dependent total pressure distributions at the aerodynamic interface plane, enabling the analysis of the flow characteristics throughout the duct length. The effect of inserting a honeycomb was tested by increasing its height from 0 to 2.2 times the baseline flow boundary layer thickness upstream of the curve. Crosstream flow symmetry was achieved for specific geometrical configurations with a negligible decrease in the pressure recovery.

4:40PM L36.00006 Experimental sensitivity analysis of a hydrodynamically self-excited low-density axisymmetric jet , LARRY LI, MATTHEW JUNIPER, University of Cambridge — We report preliminary findings from an experiment on the passive control of a hydrodynamically self-excited low-density axisymmetric jet. The control element that we use is a thin axisymmetric ring. We position this ring at different locations around the jet wavemaker and measure the response with a hot wire. We present our findings via sensitivity maps of the global frequency and the limit-cycle amplitude, and compare these to predictions from linear global instability analysis.

4:53PM L36.00007 Control of fully turbulent pipe flow , JAKOB KUEHNEN, BJOERN HOF, IST Austria — We present a novel, very simple passive control technique, where a local modification of the flow profile by means of a stationary obstacle leads to full relaminarisation downstream. Relaminarisation is achieved about 50 diameters downstream of the control point. Since, in a smooth straight pipe, the flow remains laminar from that position significant reduction in skin friction can be accomplished. High-speed stereoscopic particle image velocimetry (S-PIV) has been used to investigate and capture the development of the transitional flow downstream the obstacle. We will present S-PIV measurements as well as pressure drop measurements and videos of the development of the flow during relaminarisation. The guiding fundamental principle behind our approach to control the velocity profile will be explained and discussed.

5:06PM L36.00008 Nonlinear switched models for control of unsteady forces on a rapidly pitching airfoil[1] , SCOTT DAWSON, Princeton University, STEVEN BRUNTON, University of Washington, CLARENCE ROWLEY, Princeton University — The unsteady aerodynamic forces incident on a pitching flat plate airfoil at a Reynolds number of 100 are investigated through direct numerical simulation. Linear state-space models, identified from impulse response data via the eigensystem realization algorithm, are used to accurately track rapid changes in lift coefficient through either feedback or feedforward control, even in the presence of gust disturbances. We develop a technique to project between states of linear models obtained at different angles of attack using primal and pseudo-adjoint balanced POD modes. This allows for the formation of a nonlinear switched model that is accurate over a wide range of angles of attack, in both pre- and post-stall regimes. We additionally investigate phenomena that are not captured by linear models, such as an increase in mean lift that occurs when vortex shedding frequencies are excited. The effect of changing the pitch axis is also investigated, where it is found that pitching aft of the mid-chord results in right half plane zeros that increase the difficulty of the control problem.

[1]This work was supported by AFOSR grant FA9550-12-1-0075

5:19PM L36.00009 Transformation of steady fluid flow, in porous media, into a pulsed fluid flow; experiment findings and mathematical modeling , HASSON M. TAVOSSI, Dept., of Physics, Astronomy & Geosciences, Valdosta State University, 1500 N. Patterson St., Valdosta, GA. 31698 — In this paper we present experimentally and mathematical model for the conditions under which a steady fluid flow passing through the porous media can transform itself into a pulsed fluid flow. Our experimental findings show that a pulsed fluid flow in the porous media can result from a steady flow, under certain conditions. This paper describes experimental conditions under which such a pulsed flow can result. The experimental setup is presented and a mathematical model is obtained using analogous mechanical oscillator of a mass-spring system, and an electrical oscillator consisting of the inductor-capacitor circuit. The proposed model shows the effects of fluid parameters such as; flow-rate, pressure drop, fluid density, viscosity, pore ratio and pore shape, on the resonance frequency of this pulsed flow in the porous media.

5:32PM L36.00010 A dynamic observer to capture perturbation energy in noise amplifiers , JUAN GUZMAN, DENIS SIPP, Onera, PETER SCHMID, Ecole Polytechnique — We aim at building a reduced order model of a fluid system which accurately predicts the dynamics of a flow from a local wall measurement. This is particularly difficult in the case of noise amplifiers where the upstream noise environment triggering the receptivity of the flow is not known, which rules out classical Galerkin approaches to build reduced-order models. Here, we propose a methodology to obtain such a model from simultaneous time-resolved PIV and wall-shear stress measurements. The technique will be illustrated on the case of a transitional flat-plate boundary layer, where the snapshots of the flow are obtained with a DNS simulation. Yet, the considered approach is meant to be tractable in experiments so that special care has been taken to only use data available in an experiment. The proposed approach combines a reduction of the degrees of freedom of the system by a projection of the PIV snapshots onto a POD basis together with a system-identification technique to obtain a state-space model. Comparisons of velocity measurements at various places in the boundary layer between the DNS simulation and the obtained dynamic observer demonstrates the accuracy of the resulting model. Such a model may be used in a feedback control framework to delay transition.

Tuesday, November 26, 2013 8:00AM - 10:10AM –

Session M1 Geophysical: General II - Stratified Flows 323 - Pedram Hassanzadeh, University of California, Berkeley

8:00AM M1.00001 The Oceanic Charney Problem , SHANE KEATING, University of New South Wales, K. SHAFER SMITH, New York University — We examine the effect of a surface buoyancy gradient on the formation, vertical structure, and transport properties of mesoscale eddies in an 'oceanic' version of the classical Charney problem. The analysis is carried out in the context of a general mean state that permits a systematic study of the competing effects of surface buoyancy, planetary vorticity, and baroclinic shear. We show that the presence of a surface buoyancy gradient subtly modifies the Charney-Stern-Pedlosky necessary criteria for instability and has important implications for the resulting nonlinear equilibrated flow. In particular, a surface buoyancy gradient rapidly generates buoyancy variance close to the surface, strongly modifying the nature of the turbulent cascades, the kinetic energy spectrum, and the vertical structure of the eddies. Idealized numerical simulations of the resulting flow show a transition from surface quasigeostrophic turbulence near the surface to classical geostrophic turbulence in the interior.

8:13AM M1.00002 Global instabilities of internal gravity waves , GAÉTAN LERISSON, LadHyX, CNRS-École Polytechnique, SABINE ORTIZ, UME, ENSTA-Paristech and LadHyX, CNRS-École Polytechnique, JEAN-MARC CHOMAZ, LadHyX, CNRS-École Polytechnique — Internal gravity waves are particularly important in the ocean where they are generated by different mechanisms, interaction of currents or tides with topography, or coupling with waves at the thermocline. By their breaking they are thought to influence the deep ocean mixing and so contribute to the thermohaline circulation. We reconsider the experiment and theory of Bourget et al who considered stationary quasi-monochromatic beam to include the influence of a uniform background horizontal flow. Specifically we consider two limit cases: the non translating wave maker in which the waves are stationary and the wave maker translation at the horizontal phase velocity which to the classical lee wave problem of a sinusoidal mountain. We show that the global stability properties of these different flows differ strongly whereas locally they involve the same unstable gravity wave solution. This change in global stability is then linked to the absolute or convective nature of the local instability which are for the first time determined for a periodic base flow and for 2D wave packets.

8:26AM M1.00003 A Unified Model of Geostrophic Adjustment and Frontogenesis , JOHN TAYLOR, CALLUM SHAKESPEARE, DAMTP, University of Cambridge — Fronts, or regions with strong horizontal density gradients, are ubiquitous and dynamically important features of the ocean and atmosphere. In the ocean, fronts are associated with enhanced air-sea fluxes, turbulence, and biological productivity, while atmospheric fronts are associated with some of the most extreme weather events. Here, we describe a new mathematical framework for describing the formation of fronts, or frontogenesis. This framework unifies two classical problems in geophysical fluid dynamics, geostrophic adjustment and strain-driven frontogenesis, and provides a number of important extensions beyond previous efforts. The model solutions closely match numerical simulations during the early stages of frontogenesis, and provide a means to describe the development of turbulence at mature fronts.

8:39AM M1.00004 Turbulent mixing in stratified wall-bounded turbulent flows[1] , SUBHAS VENAYAGAMOORTHY, FARID KARIMPOUR, Colorado State University — The focus of this study is to investigate the effect of density stratification in wall-bounded turbulent flows. For a steady fully developed stably stratified channel flow, we invoke the equilibrium assumption between the turbulent kinetic energy production rate (P), dissipation rate (ϵ) of the turbulent kinetic energy (k) and the turbulent potential energy dissipation rate (ϵ_{PE}) to highlight a number of pertinent issues that have direct implications for predicting turbulent mixing. Simple formulations for the momemtum and scalar diffusivites are proposed based on the irreversible flux Richardson number and the turbulent Prandtl number. Comparisons with data of direct numerical simulation of stably stratified channel flow show remarkable agreement. These findings could be useful for modeling stratified channel flows.

[1]Funded by the National Science Foundation and the Office of Naval Research.

8:52AM M1.00005 Lilly mechanism versus Zigzag instability in the destabilisation of a stratified turbulent flow initially uniform on the vertical , JEAN-MARC CHOMAZ, LadHyX, CNRS-École Polytechnique, Palaiseau, France, CRISTOBAL ARRATIA, EPFL, Lausanne, Suisse — It is now well established that strongly stratified turbulence involves a direct inhomogeneous cascade where the vertical scale is given by the buoyancy length scale as predicted by the Billant & Chomaz (2001) scaling (BCS). But the role of the so-called zigzag instability (ZZI) in imposing this scaling remains an open question, in particular because Lilly's arguments (similar to the toroidal cascade) do not involve vertical transport as ZZI does. The argument also predicts the occurrence of vertical scales much smaller than the horizontal scale. By performing transient energy growth of perturbations around an evolving, or even turbulent, flow that is vertically uniform we demonstrate that, except for flows made of well separated vortices, the layering of the flow results from the 2D perturbation mode associated to the leading Lyapunov exponent (measuring the sensitivity to initial condition of the 2D base flow) and not from the zigzag modes coming from neutral 2D mode associated with rotation and translation. The generic route to stratified turbulence seems then to be following a Lyapunov-Lilly avenue and not the zigzag winding road. Still, no matter which mechanism involved, the BCS scaling applies to the optimal gain explaining the anisotropy of stratified turbulence.

9:05AM M1.00006 Buoyant Jets in Stratification; Mixing Efficiencies, Entropy Conditions and Wall Effects[1] , CHUNG-NAN TZOU, ROBERTO CAMASSA, MARLOW DURBIN, RICHARD MCLAUGHLIN, JEREMY WARD, COLE WHETSTONE, BRIAN WHITE, UNC Joint Fluids Lab, UNC JOINT FLUIDS LAB TEAM — An exact integral solution to the steady buoyant jet closure model in linearly stratified ambient environment is derived so that in the limit of a sharply stratified environment an entropy (nonlinear jump) condition can be established. Comparing the density evolution for the buoyant jet in the extremes of linear and sharp stratification using experiments and exact formulas, mixing efficiencies can be assessed. In turn, wall effects are explored experimentally in sharp stratification and compared to the closure theory. Lastly, the modeling of entrainment in these systems will be revisited.

[1]DMS-0502266, NSF RTG DMS-0943851, NSF RAPID CBET-1045653, NSF CMG ARC- 1025523, and NSF DMS-1009750, ONR DURIP N00014-09-1-0840

9:18AM M1.00007 Jets generated by a sphere moving vertically in stratified fluids , HIDESHI HANAZAKI, SHINYA OKINO, SHOTA NAKAMURA, SHINSAKU AKIYAMA, Kyoto University — Unsteady development of buoyant jets generated by a sphere moving vertically at constant speeds in stratified fluids is investigated. Initially, the sphere simply drags light upper fluids or isopycnal surfaces as it goes down, as long as the molecular diffusion of density is negligible. In the succeeding period, molecular diffusion of density in the boundary layer on the sphere surface becomes increasingly significant, especially in the lower hemisphere. Then, the density is no longer conserved and a vertical jet starts from the rear/upper stagnation point of the sphere, since the fluid particle of altered but small density tends to go back to its original height. Strength and radius of those jets depend significantly on stratification (Froude number), as well as the Reynolds number and the Schmidt number. These mechanisms are investigated by numerical simulations and measurements by laser induced fluorescence (LIF).

9:31AM M1.00008 3D Zombie Vortices in Rotating Stratified Shear , PHILIP MARCUS, SUYANG PEI, CHUNG-HSIANG JIANG, UC Berkeley, PEDRAM HASSANZADEH, Harvard, JOSEPH BARRANCO, SFSU, DANIEL LECOANET, UC Berkeley — We have shown that there is a finite-amplitude instability in linearly-stable, rotating, vertically-stratified, horizontally-shearing flows. The instability is due to excitations of baroclinic critical layers in which the vertical velocity of a neutrally-stable eigenmode is singular in the inviscid limit. This singularity coupled with the Coriolis and stretching terms in the vertical vorticity equation create intense vortex layers. Those layers roll-up into 3D vortices, which then de-stabilize other critical layers. These vortices, which we call *zombie* vortices, can fill the *dead zone* of a protoplanetary disk around a forming star. The vortices, either by themselves or by exciting inertio-gravity waves or acoustic waves, can transport angular momentum in a protoplanetary disk and thereby allow a protostar to form into a star. We find that the zombie vortices are similar in flows with Boussinesq, anelastic, and fully compressible equations of state. However, the rates of angular momentum transport and the mechanisms by which it is transported vary significantly in flows with different equations of state.

9:44AM M1.00009 Noise and Turbulence Generate 3D Zombie Vortices in Stably Stratified Rotating Shear Flows , SUYANG PEI, PHILIP S. MARCUS, CHUNG-HSIANG JIANG, University of California, Berkeley, PEDRAM HASSANZADEH, Harvard University, DANIEL LECOANET, University of California, Berkeley, JOSEPH A. BARRANCO, San Francisco State University — We showed previously that a linearly stable shearing, rotating, stably stratified flow has a finite-amplitude instability creating "zombie vortices" that self-replicate and fill the domain. Our flows were initialized with perturbations of one or two vortices. Our motivation was to determine whether "dead zones" in protoplanetary disks were stable, or whether they could be de-stabilized to produce vortices necessary for the final part of star formation and for planet formation. To be more relevant to astrophysics, we choose the initial conditions to be noise or turbulence with a Kolmogorov spectrum with small kinetic energy and Mach number. In a Kolmogorov spectrum, the largest eddies determine the kinetic energy and Mach number, while the smallest determine the vorticity and Rossby number $Ro \equiv \omega/f$, where ω is the vertical vorticity and f is the Coriolis parameter. The protoplanetary disks (which have large inertial ranges due to their large Reynolds numbers), can have large Rossby numbers, but weak Mach numbers and kinetic energies. It is important to know whether the triggering of the finite-amplitude instability that creates zombie vortices depends on threshold values of Mach number, kinetic energy, or the Rossby number. Here, we show it is the latter.

9:57AM M1.00010 Statistical Equilibrium and Inverse Cascades of vortical modes for rotating and stratified flows , CORENTIN HERBERT, RAFFAELE MARINO, ANNICK POUQUET, National Center for Atmospheric Research, TURBULENCE NUMERICS TEAM — Most turbulent flows appearing in nature are subjected to strong rotation and stratification. These effects break the symmetries of homogenous isotropic turbulence. In doing so, they introduce a natural decomposition of phase space in terms of wave modes and potential vorticity modes. The appearance of a new time scale associated to the propagation of waves increases the complexity of the energy transfers between the various scales; nonlinearly interacting waves may dominate at some scales while balanced motion may prevail at others. In the end, it is difficult to predict if the energy cascades downscale as in homogeneous isotropic turbulence, upscale as expected from balanced dynamics, or follows yet another phenomenology. In this contribution, we suggest a theoretical approach based on equilibrium statistical mechanics for the ideal system. We show that when the dynamics is restricted to the vortical modes, the equilibrium spectrum features an infrared divergence characteristic of an inverse cascade regime. This can be interpreted as a metastable state for the full system. We discuss how the waves are expected to deflect the energy cascade, for purely rotating, purely stratified and rotating-stratified flows, finally leading to inverse or direct cascade scenarios.

Tuesday, November 26, 2013 8:00AM - 10:10AM –

Session M2 Convection and Buoyancy-Driven Flows VI: Turbulent Convection 324 - Reza Baghaei Lakeh, University of California, Los Angeles

8:00AM M2.00001 Including APE in the energy budget of turbulent Rayleigh-Bénard convection , ROSS GRIFFITHS, BISHAKHDATTA GAYEN, GRAHAM HUGHES, Australian National University — An expanded view of the mechanical energy budget for Rayleigh-Bénard convection is developed, recognising that the available part of potential energy (APE) is the energy source for convection. Energy conversion rates and the partitioning of energy pathways between large and small scales of motion are examined using three-dimensional numerical simulations. The relative magnitudes of different pathways change significantly over the range $Ra \sim 10^7 - 10^{13}$. At $Ra < 10^7$ small-scale turbulent motions are energized directly from APE while kinetic energy is dissipated by both the large- and small-scale motions at comparable rates. In contrast, at $Ra \geq 10^{10}$ most of the APE goes into kinetic energy of the large-scale flow, which then undergoes shear instabilities sustaining small-scale turbulence. At large Ra one half of the total APE supply goes to viscous dissipation, the other half to mixing, giving a mixing efficiency of 50% as predicted theoretically. While the viscous dissipation is largely in the interior, the irreversible mixing is largely confined to the unstable boundary layers. Thus inclusion of 'the other half' of the energy in the budget provides new information on the mechanics of the interior and boundary layers, and the roles of different length scales.

8:13AM M2.00002 Turbulent plumes of unequal strength in a ventilated filling-box - thermal overshoots and bulk overturning , AJAY SHRINIVAS, Imperial College London, GARY HUNT, University of Cambridge — The activation of two non-interacting turbulent plumes of constant buoyancy fluxes B_1 and $B_2 > B_1$ in a ventilated box typically gives rise to a three-layer stratification comprised of two buoyant layers and a lower region at ambient density. A theoretical model is developed to predict the time evolution of this density stratification and the displacement flow driven by the two buoyant layers through openings, at the top and base, that connect the box to a quiescent stationary ambient of uniform density. When the top layer provides the dominant forcing, we show that the mean layer buoyancies evolve on two characteristically different timescales, thus inducing a time lag. As a result, the mean buoyancy of the intermediate (i.e. middle) layer exceeds its steady value for a significant duration, giving rise to a "thermal overshoot." This phenomenon can have key practical implications in ventilated rooms as occupants would experience "localised overheating." Furthermore, we find that the two plumes can induce a bulk overturning of the buoyant layers. We show that, for a given source strength ratio $\psi = B_1/B_2$, thermal overshoots are realised for dimensionless opening areas $A < A_{oh}$ and overturning for $A < A_{ot}$.

8:26AM M2.00003 Investigation of Transient, Turbulent Natural Convection in Vertical Tubes for Thermal Energy Storage in Supercritical CO_2[1] , REZA BAGHAEI LAKEH, ADRIENNE S. LAVINE, H. PIROUZ KAVEHPOUR, RICHARD E. WIRZ, University of California, Los Angeles — Heat transfer can be a limiting factor in the operation of thermal energy storage, including sensible heat and latent heat storage systems. Poor heat transfer between the energy storage medium and the container walls impairs the functionality of the thermal storage unit by requiring excessively long times to charge or discharge the system. In this study, the effect of turbulent, unsteady buoyancy-driven flow on heat transfer in vertical storage tubes containing supercritical CO_2 as the storage medium is investigated computationally. The heat transfer from a constant-temperature wall to the storage fluid is studied during the charge cycle. The results of this study show that turbulent natural convection dominates the heat transfer mechanism and significantly reduces the required time for charging compared to pure conduction. Changing the L/D ratio of the storage tube has a major impact on the charge time. The charge time shows a decreasing trend with Ra_L. The non-dimensional model of the problem shows that Nusselt number and non-dimensional mean temperature of the storage fluid in different configurations of the tube is a function Buoyancy-Fourier number defined as of Fo_L * Ra_L^m * L/D.

[1]This study was supported by award No. DE-AR0000140 granted by U.S. Department of Energy under Advanced Research Projects Agency - Energy (ARPA-E) and by award No. 5660021607 granted by Southern California Gas Company.

8:39AM M2.00004 Small scale anisotropy in mixed convective turbulence , HALLDÓR EINARSSON, ANDREA SCAGLIARINI, LAHCEN BOUHLALI, ARMANN GYLFASON, Reykjavik University, FEDERICO TOSCHI, Eindhoven University of Technology — Turbulent convection is present in a variety of naturally occurring flows and engineering applications. Our concern is with mixed convection, where we study the transition from the Rayleigh-Bénard (RB) convection to forced convection in a channel flow. We consider a fully developed turbulent RB cell and at a given time we apply a constant pressure gradient, orthogonal to gravity, to impose the channel flow. We will analyze some proper indicators of small scale anisotropy and focus on how they vary at changing from one regime (dominated by buoyancy) to the other (dominated by forced convection). The results are interpreted in relation to the turbulent structures present in the fluid, their timescale and size. The observations will be linked with recent experimental finding as well as previous numerical and experimental results.

8:52AM M2.00005 ABSTRACT WITHDRAWN –

9:05AM M2.00006 Nematic - isotropic phase transition in turbulent thermal convection[1] , GUENTER AHLERS, STEPHAN WEISS, Department of Physics, University of California, Santa Barbara, USA — The nematic-isotropic transition of a liquid crystal (LC) at a temperature T_{NI} is an example of a *soft* phase transition, where fluid properties, although discontinuous, change only very little and where the latent heat is small. Understanding thermal convection in the presence of such a phase change is relevant to convection in Earth's mantle where discontinuous changes of the crystalline structure occur. We report on turbulent Rayleigh-Bénard convection of a nematic LC while it undergoes a transition from the nematic to the isotropic phase in a cylindrical convection cell with aspect ratio Γ (height over diameter) of 0.50. The difference between the top- and the bottom-plate temperature $\Delta T = T_b - T_t$ was held constant, while the average temperature $T_m = (T_b + T_t)/2$ was varied. There was a significant increase of heat transport when T_{NI} was between T_b and T_t. Measurements of the temperatures along the side wall as a function of T_m showed several ranges with qualitatively different behavior of quantities such as the time-averaged side-wall temperature, temperature gradient, or temperature fluctuations. We interpret these different ranges in terms of processes in the thermal boundary layers close to the top and bottom plates.

[1]SW acknowledges support by the Deutsche Forschungsgemeinschaft. This work was supported by the U.S. National Science Foundation through Grant No. DMR11-58514.

9:18AM M2.00007 Increase of heat transfer efficiency and plume coherence induced by geometrical confinement in turbulent thermal convection[1] , KE-QING XIA, SHI-DI HUANG, MATTHIAS KACZOROWSKI, RUI NI, The Chinese University of Hong Kong — Using a classical convection model system, we show that a simple geometrical confinement can greatly increase the turbulent convective heat transfer efficiency, the Nusselt number Nu. It is found that when the aspect ratio (lateral dimension over height) of the system is decreased from 0.6 to 0.1, Nu is increased by 17% for the parameter range explored. Detailed experimental and numerical studies show that this enhancement is brought about by the changes in the dynamics and morphology of the thermal plumes in the boundary layers and in the large-scale flow structures in the bulk. It is found that the confined geometry produces more coherent and energetic hot and cold plume clusters that go up and down in random locations, resulting in more uniform and thinner thermal boundary layers. The study demonstrates how changes in turbulent bulk flow can influence the boundary layer dynamics and shows that the prevalent mode of heat transfer existing in larger aspect ratio convection cells, in which hot and cold thermal plumes are carried by the large-scale circulation along opposite sides of the sidewall, is not the most efficient way for heat transport.

[1]This work was supported by the RGC of Hong Kong SAR (CUHK403712)

9:31AM M2.00008 Test of the anomalous scaling of passive temperature fluctuations in turbulent thermal convection[1] , PENGER TONG, Department of Physics, Hong Kong University of Science and Technology, XIAOZHOU HE, Max Planck Institute for Dynamics and Self Organization, D-37073 Gottingen, Germany, XIAODONG SHANG, South China Sea Institute of Oceanology, Chinese Academy of Sciences — The scaling properties of the temperature structure function (SF) and temperature-velocity cross-structure function (CSF) are investigated in turbulent Rayleigh-Benard convection. The measured SFs and CSFs are found to exhibit good scaling in space and time and a good agreement between the CSF exponents and the thermal dissipation exponents is observed, confirming that the anomalous scaling of passive temperature fluctuations in turbulent convection is indeed caused by the spatial intermittency of the dissipation field. Furthermore, the experiment demonstrates that the functional form of the SF and CSF exponents changes with the geometry of the most dissipative structures in the flow.

[1]This work was supported by the Research Grants Council of Hong Kong SAR.

9:44AM M2.00009 Influence of thermal plumes on Lagrangian acceleration in thermally-driven turbulence[1] , XIAO-MING LI, RUI NI, SHI-DI HUANG, KE-QING XIA, The Chinese University of Hong Kong — We report an experimental study of local acceleration measurement in turbulent Rayleigh-Bénard (RB) convection. The experiment was conducted in a cylindrical cell of unity aspect ratio, spanning the range of Rayleigh number from 6.0×10^8 to 1.3×10^{10} at Prandtl number 4.3 and 6.2 respectively. We focus on the regions that are close to the thermal boundary layer and sidewall where thermal plumes dominate. The measurements were made in two tracking regions that are located 1 cm away from the sidewall and 1.5 cm above the bottom plate, respectively, both with a volume of about 5 cm^3. We find that, near the bottom thermal boundary layer, the most probable acceleration deviates from zero. This may be understood as a result of the circular motion of large-scale circulation rather than the buoyancy. We also find that, at small Ra, the acceleration variances measured at both sidewall and bottom plate show a different power law scaling and are larger than those in the cell center. As Ra increases, the variances gradually merge with those measured in the center. This result suggests that the influence of thermal plumes, or buoyancy, is significant under moderate levels of turbulent background fluctuations.

[1]This work was supported by the RGC of Hong Kong SAR (CUHK404409)

9:57AM M2.00010 Conditional temperature statistics in anisotropic turbulent thermal convection for Rayleigh numbers up to 10^{15}[1] , XIAOZHOU HE, DENNIS P.M. VAN GILS, EBERHARD BODENSCHATZ, MPIDS, Goettingen, Germany, GUENTER AHLERS, UC Santa Barbara, USA — We present systematic measurements of conditional diffusion $r(x) = \langle \ddot{X} | X = x \rangle$ and dissipation $q(x) = \langle (\dot{X})^2 | X = x \rangle$ of the normalized temperature fluctuations $X = (T - \bar{T})/\sigma$ in turbulent Rayleigh-Bénard convection (RBC) at several radial positions where the flow is anisotropic. The data cover the Rayleigh-number range $10^{13} \leq Ra \leq 10^{15}$ for a Prandtl number Pr $\simeq 0.80$. The sample was a right-circular cylinder with aspect ratio $\Gamma \equiv D/L = 0.50$ ($D = 1.12$ m is the diameter and $L = 2.24$ m is the height). We suggest analytic forms for the two conditional means and derived a general formula for the temperature probability-density function. Using $q(x)$ and $r(x)$, we calculated the normalized temperature dissipation Q.

[1]Supported by the Max Planck Society, the Volkswagenstiftung, the DFD Sonderforschungsbereich SFB963, and NSF Grant DMR11-58514.

Tuesday, November 26, 2013 8:00AM - 10:10AM –

Session M3 Multiphase Flows VII 325 - Chris Blake Ivey, Stanford University

8:00AM M3.00001 Improved volume of fluid method based on polyhedral streamtubes and embedded height functions[1] , CHRISTOPHER IVEY, PARVIZ MOIN, Stanford University, Center for Turbulence Research — A conservative advection scheme based on the use of edge-matched flux polyhedra to integrate the volume fraction evolution equation on general grids is presented. The algorithm prevents the formation of over/undershoots of the volume fraction by enforcing that the flux polyhedra do not over/underlap, removing the need for unphysical and inaccurate redistribution algorithms. Accuracy of the method derives from the edge-matched flux polyhedra's approximation of the local stream tube. Integrity of the interface representation is maintained by the use of height functions over a local cartesian stencil embedded in the mesh. Three-dimensional tests demonstrate the conservation and accuracy of the new volume of fluid method for interface tracking in general topologies.

[1]Supported by the DOE CSGF (grant number DE-FG02-97ER25308)

8:13AM M3.00002 Generalization of the Volume-of-Fluid method with realizable and planarity preserving transport of geometric moments , VINCENT LE CHENADEC, Ecole Centrale Paris - Laboratoire EM2C - UPR CNRS 288 — In two-phase flow applications, Volume-of-Fluid methods rely on a local reconstruction of the interface, performed either by approximating the interface normal based on the neighboring volume fraction values, or by designing error estimates quantifying the deviation between the exact and the reconstructed interfaces. Such estimates may be either non-local, in which case they involve neighboring volume fraction values, or local, in which case additional geometric information is required (centers of mass of each fluid, matrices of inertia,...). The latter approach presents obvious advantages in terms of computational overhead and accuracy. Transport equations for these high-order geometric moments exist, but their discretization represents a challenge, in particular when it comes to planarity preservation and realizability. In this work, we propose a geometric discretization which guarantees these properties for arbitrary moments, and its application to the first-order generalization of the Volume-of-Fluid method (Moment-of-Fluid). Within this framework, the emphasis is then set on the reconstruction, and particularly on the solution of the underlying non-linear constrained minimization. Finally, transport and reconstruction algorithms are tested in standard 2D and 3D cases.

8:26AM M3.00003 A Conservative Level Set Method on an Overset High-Resolution Cartesian Grid , MANUEL GALE, MARCUS HERRMANN, Arizona State University — Distance function level set approaches for capturing moving interfaces in multi-phase flows generally possess a major drawback: the enclosed mass is poorly conserved. Alternative methods such as the volume-of-fluid (VOF) and coupled level-set VOF provide better mass conservation, yet they face some unique challenges. Olsson and Kreiss (2005, 07) proposed a conservative level set (CLS) method that defines the level set scalar as a hyperbolic tangent away from the phase interface iso-surface. While drastically improving mass conservation, the necessary introduction of an interfacial thickness length scale, coupled to the local flow solver resolution, may adversely impact interface dynamics in complex scenarios. We propose to decouple the interface thickness scale from the local flow solver resolution scale, by solving the conservative level set scalar on an overset, high resolution Cartesian mesh, using the Refined Level Set Grid (RLSG) method. The resulting CLS-RLSG solver is coupled to a fully unstructured flow solver to solve the Navier-Stokes equations in the incompressible limit. Several test cases will be presented demonstrating the performance of the resulting code infrastructure, focusing on the interplay of local flow solver-RLSG resolution and CLS interfacial thickness.

8:39AM M3.00004 Entrainment Characteristics for variable-angle plunging liquid jets , SURAJ DESHPANDE, MARIO TRUJILLO, University of Wisconsin - Madison — Simulations based on an algebraic VoF method are used to study the entrainment characteristics of a water jet plunging into a quiescent pool at angles ranging from 10 to 90 deg. with pool. Our previous study of shallow plunging jets (Deshpande et al. 2012) revealed a discernible frequency in the formation of large air cavities. This contrasts the well-documented chaotic entrainment at steeper inclinations, suggesting a different entrainment mechanism exists for shallow angles. Quantitatively, it is found that larger cavities and greater volume of entrained air occur at shallower angles (10, 12 deg.). A precursor to the formation of these large cavities is the presence of a stagnation region in the zone of impingement. Using a local mass and momentum balance, we show that this stagnation region deflects the incoming jet at wide angles producing large air cavities. Entrainment in shallow jets is similar to the initial impact of the jet with a pool, but it occurs periodically. The recurrence is a consequence of jet disruption by traveling waves on the pool. Qualitative analysis, supported with simulations, demonstrates linear scaling of entrainment period with Froude number.

8:52AM M3.00005 The effect of residence time on the dynamics of a condensating aerosol in a Hiemenz-type stagnation flow , AMJAD ALSHAARAWI, KUN ZHOU, GIANFRANCO SCRIBANO, ANTONIO ATTILI, FABRIZIO BISETTI, King Abdullah University of Science and Technology, CLEAN COMBUSTION RESEARCH CENTER TEAM — The effect of residence time on the formation and growth of a condensating aerosol is simulated in a Hiemenz-type stagnation flow setup, for which a unique and well-defined time scale characterizes the velocity field. In this configuration, a hot stream saturated with dibutyle phthalate (DBP) vapor mixes with a cold dry stream. A mixing layer forms at the stagnation plane triggering supersaturation and droplets are generated by homogeneous nucleation. Aerosol dynamics are simulated using the Quadrature Method of Moments (QMOM). Two regimes related to the flow residence time are observed, i.e., a nucleation regime and a condensation regime. The nucleation regime, at short residence times, is characterized by the consumption of DBP vapor into droplets having a negligible effect on the vapor phase. In this regime, both the number density and volume fraction of droplets increase with residence time. In the condensation regime, at long residence times, vapor condensation consumes the vapor phase considerably. For longer residence times, more vapor is consumed, resulting in lower number densities due to the lower nucleation rates, whereas the volume fraction saturates.

9:05AM M3.00006 Three-dimensional advected normals method for calculating interfacial normals and curvatures in two-phase flows , ASHISH PATHAK, MEHDI RAESSI, University of Massachusetts Dartmouth — We present an extension of the advected normals method to three-dimensional two-phase flows, including contact line problems. In this method, a mass-conserving volume-of-fluid method is used to track fluid volumes, while the unit vectors normal to the fluid interfaces are advected by solving an additional transport equation. Interface curvature is computed directly from the advected normals. RK3 scheme is used for discretizing the temporal gradient of the normals transport PDE, and spatial gradients are calculated using Lax-Friedrichs flux splitting scheme with WENO-5, which provides a more robust solution, especially in cases where the velocity field may contain spurious currents. Efficacy of the method in accurate evolution of normals is demonstrated in 3D test cases with prescribed velocity, where the normals and curvatures are shown to converge with second and first order accuracy, respectively. Furthermore, the method was extended to handle contact line problems in 3D. Normal vectors around the contact line and along the contact surface are used as boundary conditions to impose the contact angle. Additionally, to solve the normals evolution PDE, an extension of the normals field below the contact surface is required, which is obtained using natural neighbor interpolation.

9:18AM M3.00007 Unifying binary fluid diffuse-interface models in the sharp-interface limit , DAVID SIBLEY, ANDREAS NOLD, SERAFIM KALLIADASIS, Department of Chemical Engineering, Imperial College London, UK — Flows involving free boundaries occur widely in both nature and technological applications, existing at liquid-gas interfaces (e.g. between liquid water and water vapour) or between different immiscible fluids (e.g. oil and water, and termed a binary fluid). To understand the asymptotic behaviour near a contact line, a liquid-gas diffuse-interface model has been investigated recently [1]. In contrast, here we investigate the behaviour between two ostensibly immiscible fluids, a binary fluid, using related models where the interface has a thin but finite thickness. Quantities such as the mass fraction of the two fluid components are modelled as varying smoothly but rapidly in the interfacial region. There has been a wide variety of models used for this situation, based on Cahn–Hilliard or Allen–Cahn theories coupled to hydrodynamic equations, and we consider the effect of these differences using matched asymptotic methods in the important sharp-interface limit, where the interface thickness goes to zero. Our aim is to understand which models represent better the classical hydrodynamic model and associated free-surface boundary conditions.

[1] Sibley, Nold, Savva, Kalliadasis. Eur. Phys. J. E 36, 26 (2013)

9:31AM M3.00008 Linear stability analysis of miscible two-fluid flow in a channel with velocity slip at the walls , SUKHENDU GHOSH, R. USHA, Department of Mathematics, IIT Madras, Chennai-600036, India, KIRTI CHANDRA SAHU, Department of Chemical Engineering, IIT Hyderabad, Yeddumailaram - 502205, Andhra Pradesh, India — The linear stability characteristics of pressure-driven three-layer flow of two miscible liquids with same density and varying viscosity in a channel with velocity slip at the wall are examined.The flow system is destabilizing when a more viscous fluid occupies the region closer to the wall with slip. For this configuration, a new mode of instability,namely,the overlap mode, similar to the one which appears in the corresponding flow in a rigid channel, occurs for high mass diffusivity of the two fluids,when the critical layer of the disturbance overlaps the viscosity stratified layer.The co-existence of several overlap modes, TS mode are also observed under certain circumstances.The flow is unstable at low Reynolds numbers for a wide range of wave numbers for low mass diffusivity.A configuration with less viscous fluid adjacent to the wall is stabilizing at moderate miscibility. It is possible to achieve stabilization or destabilization of miscible two-fluid flow in a channel with wall slip by appropriately choosing the viscosity of the fluid layer adjacent to the wall. In addition, the velocity slip at the wall has a dual role in stabilizing the flow system.The flow system can be either stabilized or destabilized by designing the walls of the channel as hydrophobic surfaces.

9:44AM M3.00009 Three-dimensional simulations of pressure-driven displacement flow of two immiscible liquids using a multiphase Lattice Boltzmann approach , PRASANNA R. REDAPANGU, KIRTI CHANDRA SAHU, Indian Institute of Technology Hyderabad, S.P. VANKA, University of Illinois at Urbana-Champaign — A three-dimensional multiphase lattice Boltzmann approach is used to study the pressure-driven displacement flow of two immiscible liquids of different densities and viscosities in an inclined square duct. A three-dimensional-fifteen-velocity (D3Q15) lattice model is used. The simulations are performed on a graphics processing unit (GPU) based machine. The effects of channel inclination, viscosity and density contrasts are investigated. The contours of the density and the average viscosity profiles in different planes are plotted and compared with two dimensional simulations. We demonstrate that the flow dynamics in three-dimensional channel is quite different as compared to that of two-dimensional channel. In particular, we found that the flow is relatively more coherent in three-dimensional channel than that in two-dimensional channel. A new screw-type instability is seen in the three-dimensional channel which cannot be observed in two-dimensional simulations.

9:57AM M3.00010 Analysis of the two-phase flow in the wake of a transom stern[1] , KELLI HENDRICKSON, Massachusetts Institute of Technology - MIT, GABRIEL WEYMOUTH, University of Southampton, DICK YUE, Massachusetts Institute of Technology - MIT — The objective of this effort is to understand the physical and air entrainment characteristics of the two-phase flow in the wake of a transom stern. High-resolution numerical simulations are performed on the wake of a canonical transom stern at large scales using conservative Volume-of-Fluid (cVOF) and implicit Large Eddy Simulation (iLES). Boundary Data Immersion Method (BDIM) is used to simulate the dry transom stern wake region at three different Froude numbers and two different effective viscosities. A novel Lagrangian cavity identification algorithm based on computer graphics techniques enables the analysis of the temporal evolution of the entrained air cavities. Analysis of the simulation results for the flow structure and air entrainment of the large air cavities will be presented, including the scaling with ship parameters.

[1]Supported by the Office of Naval Research

Tuesday, November 26, 2013 8:00AM - 9:57AM –

Session M4 Turbulence: Modeling IV 326 - Sanjiva K. Lele, Stanford University

8:00AM M4.00001 A subfilter-scale stress model for large eddy simulations , AMIRREZA ROUHI, UGO PIOMELLI, Queen's University — In most large eddy simulations, the filter width is related to the grid. This method of specification, however, causes problems in complex flows where local refinement results in grid discontinuities. Following the work of Piomelli and Geurts (*Proce. 8th Workshop on DLES*, 2010) we propose an eddy-viscosity approach in which the filter width is based on the flow parameters only, with no explicit relationship to the grid size. This model can achieve grid-independent LES solutions, vanishing dynamically in the regions of low turbulence activity and a computational cost less than the dynamic models. The Successive Inverse Polynomial Interpolation (Geurts & Meyers *Phys. Fluids* **18**, 2006) was used to calculate the model parameter. Calculating implicitly the eddy-viscosity at each time-step removes the numerical instabilities found in previous studies, while maintaining the local character of the model. Results of simulations of channel flow at Re_τ up to 2,000, and forced homogeneous isotropic turbulence will be presented.

8:13AM M4.00002 Grid-independent large-eddy simulation (LES) of turbulent flow around a circular cylinder using explicit filtering , SATBIR SINGH, DONGHYUN YOU, Carnegie Mellon University — The explicit filtering technique has the potential to provide grid-independent and error-quantified large-eddy-simulation (LES) solutions. Bose *et al.* [Phys. Fluids **22**, 105103 (2010)] and Singh *et al.* [Phys. Fluids **24**, 085105 (2012)] recently obtained grid-independent LES solutions for turbulent channel flow using one-dimensional discrete filter functions implemented on Cartesian grids. Many complex flow configurations, however, employ arbitrary shape grids, for which it is difficult to design such discrete filter functions. In the present work, we employ an elliptic differential filter to solve explicit-filter LES equations on arbitrary shaped grids. The coefficients of the elliptic filter are determined by comparing its filtering characteristics with those of a Gausian filter. The elliptic filter is applied to a homogeneous isotropic turbulence flow field and the coefficient is adjusted until a filtered energy spectra similar to that of the Gaussian filter is obtained. The filter coefficients thus obtained are then employed to solve explicit-filter LES equations for turbulent channel flow at $Re_\tau = 395$ and turbulent flow over a circular cylinder at $Re_D = 3900$. Grid-independent solutions are obtained for both flow configurations.

8:26AM M4.00003 LES of three-dimensional, shear-driven turbulent wall flow at $Re_\tau \approx 2000$ using a nested-LES wall-modeling approach , YIFENG TANG, RAYHANEH AKHAVAN, The University of Michigan, Ann Arbor, MI 48109-2125 — Accurate prediction of high Reynolds number non-equilibrium wall flows presents a major challenge for traditional LES and wall modelling approaches such as hybrid RANS/LES or analytical wall functions. In this study, we investigate the applicability of the nested-LES wall-modeling approach (Tang & Akhavan 2012) to non-equilibrium flow in a 3D turbulent channel at $Re_\tau \approx 2000$. The three-dimensionality was introduced by imposing an impulsive spanwise motion of the walls in an initially 2D equilibrium turbulent channel flow and suddenly stopping the spanwise motion after the turbulence had adjusted to the wall motion. The progression of turbulence statistics in both the sheared and recovery stages was in good agreement with experiments in 3D, shear-driven boundary layers at comparable Reynolds number (Driver & Hebbar 1987). The nested-LES wall-modeling approach couples coarse-grained LES in a full-size domain ($L_x = 4\pi h, L_y = 2\pi h$) with nested fine-grained LES in a minimal domain ($L_x^+ = 3200, L_y^+ = 1600$), both using 64^3 grids. At each iteration, the velocity fields in both domains are renormalized to match the U_i and $u_{i,rms}$ to those of the minimal domain in the near-wall region and the full-size domain in the outer region, respectively.

8:39AM M4.00004 Turbulence Shell Models for Initial and Inflow Conditions in Direct and Large-Eddy Simulations , TOMASZ DROZDA, JEFFERY WHITE, ROBERT RUBINSTEIN, NASA Langley Research Center — Initial and inflow conditions continue to present a challenge for simulations of turbulent flows via Direct and Large-Eddy Simulations (DNS and LES). The current work utilizes the output of a Sabra [1] shell model of turbulence to synthesize a three-dimensional (3D) homogeneous, isotropic, incompressible, turbulence-in-a-box velocity field. This approach is motivated by recent work of van de Water et al. [2] on generation of wind tunnel turbulence with active grids. The properties of the new synthetic turbulence are assessed for several values of the Reynolds number by computing higher order statistics. DNS of the decay of homogeneous isotropic turbulence are also considered with initial conditions obtained using both the new method and Gaussian turbulence.

[1] L'vov, V., Podivilov, E., Pomyalov, A., Procaccia, I., and Vandembroucq, D., Improved shell model of turbulence, *Phys. Rev. E.*, **58**:1811–1822, 1998.
[2] van de Water, W., Cekli, H.E., and Joosten, R., Stirring turbulence with turbulence, in *Bulletin of the American Physical Society*, the American Physical Society, Baltimore, MD, 2011.

8:52AM M4.00005 Subgrid model evaluation through lockstep DNS/LES of a turbulent jet , ANKIT BHAGATWALA, Sandia National Laboratories, VENKAT RAMAN, University of Texas Austin, JACQUELINE CHEN, Sandia National Laboratories — The aim of this study is to analyze the validity of the common Smagorinsky type closure assumptions employed in LES scalar mixing and scalar dissipation rate models. This is done using a unique DNS-LES lockstep methodology, wherein a DNS is run simultaneously with several LES instances. The LES only solves for the scalar fields and obtains the velocity fields directly from the filtered DNS solution at every substep of time. The LES is also solved on the same grid as the DNS. This eliminates two primary sources of error in LES, numerical error associated with a coarser grid and modeling error arising from the modeled velocity field. The only source of error then, is from the closure assumption made for the LES model. One instance of DNS and three LES instances of a 3D turbulent slot jet at a Reynolds number of 7500 are simulated. The three LES simulations correspond to three different filter widths. Predictions of resolved and subgrid contributions of scalar second moment, scalar variance and scalar dissipation rate are compared. Implications for turbulent combustion models that heavily rely on these parameters are discussed.

9:05AM M4.00006 Experimental study of the SGS pressure-strain-rate correlation in the convective atmospheric surface layer[1] , KHUONG NGUYEN, CHENNING TONG, Clemson University — The subgrid-scale (SGS) stress and flux are studied using measurement data obtained in the atmospheric surface layer during the Advection Horizontal Array Turbulence Study (AHATS) field program, which notably includes measurement of the resolvable- and subgrid-scale pressure. We analyze the terms in the transport equations of the SGS stress and SGS heat flux, conditioned on the resolvable-scale velocity, for different filter scales and atmospheric stability. The results show that the pressure destruction terms in the budgets of the SGS shear stress and the SGS heat flux play the usual role of return-to-isotropy and generally counter the trends of the conditional production for all filter scales and unstable conditions. In contrast, the pressure-strain-rate correlations in the budgets of the normal SGS stress components can be the main cause of anisotropy of the SGS stress under convective conditions, depending strongly on the resolvable-scale velocity. These effects are most significant at large filter scales and have strong implications for modeling the near-wall SGS pressure-strain-rate correlation.

[1]Supported by NSF

9:18AM M4.00007 Higher-order moments and their modeling approximations in turbulent channel flow , ELBERT JEYAPAUL, National Institute of Aerospace, GARY COLEMAN, NASA Langley Research Center — Third- and fourth-order moments and the terms in their budgets are evaluated using results from Direct Numerical Simulations (DNS) of turbulent channel flow at $Re_\tau = 395$, to aid in the development of higher-order Reynolds-Averaged Navier Stokes (RANS) closure models. These models have been proposed as a means of obtaining more accurate predictions of complex flows. The DNS data is used to test the assumptions that have been made to model the turbulent diffusion, velocity-pressure gradient and dissipation terms in the higher-order transport equations. The validity of using the Gram-Charlier Probability Density Function (PDF) to extrapolate the fourth-order moments from the lower-order ones is examined, as is the Millionshchikov hypothesis of quasi-normal distributions of the fourth-order moments. The wall-correction-free velocity-pressure gradient model of Poroseva (2001) is assessed, along with the assumption for wall-bounded flows of zero-dissipation in the third- and fourth-order equations.

9:31AM M4.00008 Stochastic model representation of the energy transfers in turbulent channel flow[1] , VASSILI KITSIOS, Laboratory For Turbulence Research in Aerospace and Combustion, Monash University, Clayton, Australia, JUAN A. SILLERO, School of Aeronautics, Universidad Politécnica de Madrid, Madrid, Spain, JULIO SORIA, Laboratory For Turbulence Research in Aerospace and Combustion, Monash University, Clayton, Australia, JORGEN S. FREDERIKSEN, CSIRO Marine and Atmospheric Research, Aspendale, Australia — A stochastic model is used to represent the energy transfers in the direct numerical simulation (DNS) of turbulent channel flow. The DNS has a Fourier discretisation in the streamwise and spanwise directions, and a Chebyshev discretisation in the wall normal direction. We spectrally decompose the DNS into large and small horizontal scales, and develop a stochastic subgrid model representing the effect the removal of the small scales has on the large. The stochastic model consists of a deterministic drain dissipation acting on the resolved field and a stochastic backscatter force. Positive values of the drain operator indicate energy sent from the large to the small scales (dissipation), whilst negative values represent energy sent from the small to the large (deterministic backscatter). The variance of the stochastic force quantifies the extent to which the backscatter is random as opposed to deterministic. We are also able to produce large eddy simulations using this stochastic subgrid model that reproduces the time averaged kinetic energy spectra of the DNS within the resolved scales. Results are presented for various Reynolds numbers up to $Re_\tau = 950$.

[1]Funded by ERC

9:44AM M4.00009 Formulation of a $k-\omega$ based DDES model , KARTHIK RUDRA REDDY, PAUL DURBIN, Iowa State University — DES models fall under the category of Hybrid RANS/LES models, and they employ RANS to resolve near-wall boundary layers in the flow domain and LES away from the surface. The idea of DES is applicable to any RANS model, and various versions of the method reflect this. A general DDES formulation was put forth (Spalart et al, 2006) suitable for use with any RANS model, and later adapted for use with the $k-\omega$ SST model (Gritskevich et al, 2012). The current work develops a variant based on the $k-\omega$ model. In this version, length scales enter directly in the subgrid eddy-viscosity, rather than being used in the dissipation term of the k-equation; indeed, one can approach it as replacing length-scale clipping, in standard DDES, with velocity scale clipping. The length scales were modified in order to account for the log-layer mismatch, a well-known issue with DDES. Simulation results from channel flow and flow over a backward-facing step are presented.

Tuesday, November 26, 2013 8:00AM - 9:57AM –

Session M5 CFD VII: Numerical Methods I 327 - Daniel Haworth, Pennsylvania State University

8:00AM M5.00001 Using Adjoint-Based Approach to Understand Flapping-Wing Aerodynamics[1] , MIN XU, MINGJUN WEI, New Mexico State University — The study of flapping-wing aerodynamics is a problem with very large control space. Adjoint-based approach, by solving an inverse problem, can be used here as an efficient tool for optimization and physical understanding. However, the adjoint equation is typically formulated in a fixed domain. The moving boundary or morphing domain brings in an inconsistency in the definition of arbitrary perturbation at the boundary, which then proposes a new challenge if the control parameters happen to be also at the boundary. An unsteady mapping function, as a usual remedy for such problems, would make the whole formulation too complex to be feasible. Instead, we use non-cylindrical calculus to re-define the perturbation and solve the inconsistency problem caused by moving/morphing solid boundaries. The approach is first validated for a simple case of a plate plunging in an incoming flow. Then we extend the approach to drag reduction and efficiency improvement of more complex cases. The optimized parameters provide a unique opportunity for physical understanding by comparison to the initial parameters.

[1]Supported by AFOSR

8:13AM M5.00002 ABSTRACT WITHDRAWN –

8:26AM M5.00003 ABSTRACT WITHDRAWN –

8:39AM M5.00004 A Monolithic Algorithm for High Reynolds Number Fluid-Structure Interaction Simulations[1] , ERIKA LIEBERKNECHT, JASON SHELDON, JONATHAN PITT, The Pennsylvania State University — Simulations of fluid-structure interaction problems with high Reynolds number flows are typically approached with partitioned algorithms that leverage the robustness of traditional finite volume method based CFD techniques for flows of this nature. However, such partitioned algorithms are subject to many sub-iterations per simulation time-step, which substantially increases the computational cost when a tightly coupled solution is desired. To address this issue, we present a finite element method based monolithic algorithm for fluid-structure interaction problems with high Reynolds number flow. The use of a monolithic algorithm will potentially reduce the computational cost during each time-step, but requires that all of the governing equations be simultaneously cast in a single Arbitrary Lagrangian-Eulerian (ALE) frame of reference and subjected to the same discretization strategy. The formulation for the fluid solution is stabilized by implementing a Streamline Upwind Galerkin (SUPG) method, and a projection method for equal order interpolation of all of the solution unknowns; numerical and programming details are discussed. Preliminary convergence studies and numerical investigations are presented, to demonstrate the algorithm's robustness and performance.

[1]The authors acknowledge support for this project from the Applied Research Laboratory Eric Walker Graduate Fellowship Program.

8:52AM M5.00005 Partitioned fluid-structure interaction scheme for bodies with high flexibility , TIMOTHY FITZGERALD, University of Maryland College Park, MARCOS VANELLA, ELIAS BALARAS, George Washington Unversity, BALAKUMAR BALACHANDRAN, University of Maryland College Park — There are many interesting problems involving fluid-structure interaction (FSI) systems such as flapping wings in micro-air-vehicles. In order to better understand these systems, high-fidelity simulation tools are needed to do the following: (i) fully capture the physics and (ii) provide a basis to construct low-fidelity models used in design. Here, a novel FSI strategy is introduced, through which a large scale fluids solver is combined with a solver for solids with high flexibility. The Navier-Stokes equations for incompressible flow are discretized by using standard central finite differences on a staggered mesh. The fluid domain is spatially decomposed through the use of the FLASH modeling framework. The solid body is discretized via geometrically exact Total Lagrangian finite elements. A novel hyperelastic material law that extends the engineering stress-strain law to finite deformations and arbitrary rotations is also implemented. The Lagrangian body is embedded in the Cartesian fluid grid by immersed boundary methods. The time marching predictor-corrector coupling procedure is based on the use of Adams methods for the fluid and the Generalized-α method for the body. We will present examples of flexible oscillating plates and a flapping Manduca Sexta wing.

9:05AM M5.00006 Richardson Extrapolation using DNAD , ISMAIL B. CELIK, HAYRI SEZER, SURYANARAYANA R. PAKALAPATI, Department of Mechanical and Aerospace Engineering, West Virginia University, P.O. Box 6106, Morgantown, WV, USA, WVU-CFD TEAM — Dual Number Automatic Derivation (DNAD) is a technique whereby a computer code can be executed with additional variable declarations to extend real number to a two dimensional space which is then used to evaluate derivatives to machine accuracy. In the literature this technique is usually applied to study sensitivities of calculations to model parameters, but not the mesh size. The current study explores possibilities of using the same technique to evaluate the derivative of the numerical solution with respect to mesh size which in turn can be used in the Taylor series expansion of the discretization error to calculate the error itself by way of extrapolation. Thus the new method enables explicit Richardson extrapolation by using only one set of calculations on a single grid. The extrapolation can be improved if an additional set of calculations are performed on a finer or a coarser mesh. The concept is demonstrated using one-dimensional example problems. Possible extension to multi-dimensions is discussed.

9:18AM M5.00007 A balanced-force finite-element method for surface-tension-driven interfacial flows using interface-capturing approaches , ZHIHUA XIE, DIMITRIOS PAVLIDIS, JAMES PERCIVAL, Imperial College London, JEFFERSON GOMES, University of Aberdeen, CHRISTOPHER PAIN, OMAR MATAR, Imperial College London — Interfacial flows with surface tension are often found in industrial and practical engineering applications, including bubbles, droplets, liquid film and jets. Accurate modelling of such flows is challenging due to their highly complex dynamics, which often involve changes of interfacial topology. We present a balanced-force finite-element method with adaptive unstructured meshes for interfacial flows. The method uses a mixed control-volume and finite element formulation, which ensures the surface tension forces, and the resulting pressure gradients, are exactly balanced, minimising the spurious velocities often found in numerical simulations of such flows. A volume-of-fluid-type method is employed for interface capturing based on a compressive control-volume advection method, and second-order finite element methods. A distance function is reconstructed from the volume fraction on the unstructured meshes, which provides accurate estimation of the curvature. Numerical examples of an equilibrium drop and dynamics of bubbles (droplets) are presented to demonstrate the capability of this method.

9:31AM M5.00008 Application of Kelvin's Inversion Theorem in mesh based numerical simulation of flows in unbounded domains , JOHN RUSSELL, Professor Emeritus, Florida Institute of Technology — One may decompose an unbounded domain exterior to a solid body into two parts separated by a sphere of radius a, which I will call the *Reflecting Sphere*. The *Near Exterior* is exterior to the solid body but interior to the Reflecting Sphere while the *Far Exterior* is exterior to both. Suppose the velocity field in the Far Exterior has a velocity potential, ϕ. In the 1840s KELVIN showed that the change of position variable $\mathbf{r} \to \mathbf{q}$ defined by $\mathbf{r}/r = \mathbf{q}/q$ and $rq = a^2$ (in which $r = |\mathbf{r}|$ and $q = |\mathbf{q}|$) maps the Far Exterior to an *Inverted Far Exterior* (the interior of the Reflecting Sphere). Furthermore, if $\mathbf{r} \mapsto \phi$ is a solution of LAPLACE's equation in the $\mathbf{r}$-coordinates then $\mathbf{q} \mapsto \Phi$, in which $\phi = (q/a)\Phi$, is a solution of LAPLACE's equation in the $\mathbf{q}$-coordinates. Boundedness of both the Near Exterior and Inverted Far Exterior enables simultaneous solution of the relevant partial differential equations provided one applies suitable compatibility conditions on the Reflecting Sphere. The talk will present simulations of this kind along with comparisons with analytical solutions.

9:44AM M5.00009 Smoothed Particle Hydrodynamics Continuous Boundary Force method for Navier-Stokes equations subject to a Robin boundary condition[1] , WENXIAO PAN, JIE BAO, ALEXANDRE TARTAKOVSKY, Pacific Northwest National Laboratory — A Continuous Boundary Force (CBF) method was developed for implementing Robin (Navier) boundary condition (BC) that can describe no-slip or slip conditions (slip length from zero to infinity) at the fluid-solid interface. In the CBF method the Robin BC is replaced by a homogeneous Neumann BC and an additional volumetric source term in the governing momentum equation. The formulation is derived based on an approximation of the sharp boundary with a diffuse interface of finite thickness, across which the BC is reformulated by means of a smoothed characteristic function. The CBF method is easy to be implemented in Lagrangian particle-based methods. We first implemented it in smoothed particle hydrodynamics (SPH) to solve numerically the Navier-Stokes equations subject to spatial-independent or dependent Robin BC in two and three dimensions. The numerical accuracy and convergence is examined through comparisons with the corresponding finite difference or finite element solutions. The CBF method is further implemented in smoothed dissipative particle dynamics (SDPD), a mesoscale scheme, for modeling slip flows commonly existent in micro/nano channels and microfluidic devices.

[1]The authors acknowledge the funding support by the ASCR Program of the Office of Science, U.S. Department of Energy.

Tuesday, November 26, 2013 8:00AM - 10:10AM –

Session M6 Microfluids: Electrokinetics 328 - Ali Mani, Stanford University

8:00AM M6.00001 High Order WENO Simulation of Electrokinetic Instability in a Cross-Shaped Microchannel , QIAN LI, YANN DELORME, STEVEN FRANKEL, Purdue University — Electroosmotic flow with electrokinetic effects is the primary method of fluid handling in micro-total analysis systems. Knowledge of electrokinetic instabilities (EKI) is required to trigger instabilities in applications like low Reynolds number micromixing or to suppress them in applications such as sample injection, separation and controlled diffusion-limited reaction processes where the minimum sample dispersion is needed. A novel multiblock high order in-house solver based on WENO scheme is applied to simulate the EKI for multiple electrolyte solutions with different electric conductivities in a cross-shaped microchannel. 3D simulations are performed to explore the effects of variations of applied electric field, electric field ratio, and conductivity ratios on the EKI phenomena, and to determine the critical value of electric field required for instabilities. The validity of the numerical study is assessed by comparing the numerical results with the experimental data.

8:13AM M6.00002 Electro-osmotic Flow over a Charged Super-hydrophobic Surface , HUI ZHAO, University of Nevada Las Vegas — A super-hydrophobic surface has a large effective hydrodynamic slip length compared to a smooth hydrophobic surface and holds the promise of enhancing electrokinetic flows that find many applications in microfluidics. However, recent theoretical studies suggested that electro-osmotic flows over a weakly charged, super-hydrophobic surface can only be enhanced when liquid-gas interfaces are charged. So far there is little work reported when the zeta potential of the surface is comparable or even larger than the thermal potential. Here we numerically investigate electro-osmotic flows over a periodically striped slip-stick surface by solving the standard Poisson-Nernst-Planck equations. Our results indicate that at large zeta potentials, even if liquid-gas interfaces are charged, the non-uniform surface conduction due to the mismatch between surface conductions over no-shear and no-slip regions leads to electric field lines penetrating the double layer and thus the non-uniform surface conduction weakens the tangential component of the electric field which primarily drives electro-osmotic flows. Our results imply that in the presence of strong non-uniform surface conduction, enhanced electro-osmotic flows over a super-hydrophobic surface are possible only in certain conditions. In particular, the enhancement due to the slip can potentially be lost at large zeta potentials.

8:26AM M6.00003 Modification of the local electric field around a sharp corner due to surface conductance , HSIEN-HUNG WEI, National Cheng Kung University, DAVID HALPERN, University of Alabama — It is well known that the electric field at the tip of an insulated wedge is singular when solving the two-dimensional Laplace equation for the electric potential. But if a wedge is highly charged to possess strong electric currents along the wedge surface, an imbalance of these currents can produce the so-called surface conductance effects that can either draw the electric field lines into or out of the wedge surface, and hence modify the local electric field behavior around the tip. We find that how an external field is applied is crucial to how surface conductance impacts the corner field, depending on if the applied field cuts around the wedge (cutting mode) or acts symmetrically over the wedge (impinging mode). For each mode, we not only examine how the field around the wedge behaves as the strength of surface conductance varies, but also address whether the singularity at the tip is enhanced/relieved by identifying how the field grows/decreases with distance from the tip.

8:39AM M6.00004 Electrokinetic instability of isotachophoresis shocks , GIANCARLO GARCIA, JUAN SANTIAGO, ALI MANI, Department of Mechanical Engineering, Stanford University — Isotachophoresis (ITP) is an electrokinetic focusing technique used in a variety of life science and analytical chemistry applications. In ITP, an electrokinetic shock wave forms at the interface between leading and trailing electrolytes with relatively high and low conductivities. The ITP interface is self-sharpening, as restoring electromigration fluxes counteract molecular diffusion. However, the large electric field gradient at the shock interface also gives rise to free charge and strong electrostatic body forces. At large applied currents, electrostatic forces cause recirculating flows which destabilize the ITP interface. We performed stability analysis and direct simulation of ITP shocks through numerical solutions to the coupled Nernst-Planck and Navier-Stokes equations using a quasi-electroneutral approximation. In both experiments and numerical simulations, we observe two modes of instability: 1) a distorted ITP interface which is steady in time, and 2) an oscillating perturbation which persists. In addition, at the highest simulated electric fields, we observe transition towards more chaotic oscillatory modes. We use our stability analysis and numerical simulations to characterize instability of ITP shocks using two dimensionless parameters.

8:52AM M6.00005 Electrokinetic instability and hydrodynamic chaos near electrodes , SCOTT M. DAVIDSON, MATHIAS B. ANDERSEN, ALI MANI, Stanford University — It is known that ion-concentration-polarization (ICP) near ion-selective membranes can lead to electrokinetic instability of an aqueous solution. Consistent with experimental observations, recent DNS studies demonstrate these instabilities and even predict hydrodynamic chaos when ICP is subject to high voltage. Through direct numerical simulation (DNS) of the coupled Poisson-Nernst-Planck and Navier-Stokes equations in two dimensions, we demonstrate that this phenomena is not limited to membranes, but is much more general. Our DNS results predict sustained chaotic behavior between blocking parallel electrodes under applied AC forcing and at an ideally polarizable cylinder in a DC electric field. Comparison with asymptotic predictions in the linear, nonlinear, and chaotic regimes is performed as well as analysis of transport effects.

9:05AM M6.00006 Streaming potential and conductivity measurements reveal electrokinetic properties of porous and charged layers , ALEXANDER BARBATI, BRIAN KIRBY, Cornell University — We perform streaming potential, conductivity, and supporting physical and chemical measurements on thin Nafion polymer films spun on rigid glass slides. Our data reveals a phenomenological zeta potential that scales inversely with the negative logarithm of ionic strength (electrolytes NaCl and HCl) and displays a weak, but unexpected, dependence on pH. Using derived coupling coefficients for streaming current and conductivity, we analyze the phenomenological zeta potential to extract porous layer resistance, fixed charge density, and the Donnan potential within the porous layer. We supplement these electrokinetic studies with physical and chemical measurements of the sample, using profilometry, XPS, and ellipsometry measurements to further inform the state of the system.

9:18AM M6.00007 Shear Flow induced Electrical Current Generation , CLAUS-DIETER OHL, SILVESTRE ROBERTO GONZALEZ AVILA, CHAOLONG SONG, LUONG TRUNG DUNG, Nanyang Technological University — Electro-osmotic flows are driven by an electric potential difference along a channel where the driving force is acting very close to the boundary at the electric double layer (EDL). The charge separation within the EDL gives rise to an electric current. Conversely, one may expect that a strong shear flow can induce an electric current that could be picked up with electrodes and a closed circuit. Previous experiments relied on a steady free jet at a nozzle exit driven by a strong pressure gradient [1]. Here we utilize a laser induced cavitation bubble near an electrode equipped surface to generate strong shear from the impinging jet. Correlation of high-speed recordings of the spreading jet with current measurements reveals that the shear stress is causing the electric current. We make an attempt to calibrate this sensor in a better defined shear flow within a microfluidic channel.

[1] A.M. Duffin and R.J. Saykally, "Electrokinetic Power Generation from Liquid Water Microjets," J. Phys. Chem. C 112, 17018-17022 (2008).

9:31AM M6.00008 Probing electrokinetics in microchannels and nanochannels with electrochemical measurements[1] , JARROD SCHIFFBAUER, SINWOOK PARK, GILAD YOSSIFON, Technion, Israel Institute of Technology, Faculty of Mechanical Engineering, Miro-Nanofluidics Laboratory — We present a brief review of recent experimental and theoretical results concerning the use of electrochemical impedance spectroscopy (EIS,) in conjunction with other electrochemical measurements (chronoamperometry, linear sweep voltammetry,) to characterize the response of micro- and nanofluidic systems. Using these techniques, the interplay between conduction, diffusion, and convection are probed across a range of time- and length scales. The resulting information permits characterization of the respective roles of processes in both micro- and nanchannel regions of a fluidic device. Such techniques provide a useful probe of transient behavior at the micro-nanochannel interface, have great potential in biomolecular sensing applications, and may be useful in the study of surface properties at the fluid-solid interface.

[1]We wish to acknowledge Israel Science Foundation, grant number 2015240, the Technion Russel-Berrie Nanotechnology Institute (RBNI) and a fellowship grant from the Techion Faculty of Mechanical Engineering.

9:44AM M6.00009 Geometric Modulation of Electro-Osmosis of the Second Kind in Micro-Nanochannel Interface Devices , GILAD YOSSIFON, NETA LEIBOWITZ, YOAV GREEN, JARROD SCHIFFBAUER, SINWOOK PARK, Technion - Israel Institute of Technology — The charge-selective ionic transport through the nanochannel induces a concentration polarization effect. At sufficiently high currents, the depleted region develops an extended space charge layer adjacent to the micro-nanochannel interface. As the applied voltage exceeds a critical threshold, the loss of mechanical stability in this space-charge region leads to the formation of fast fluid vortices which undergo a complex wavelength-selection process. Both microchannel dimensions and interfacial geometry have been shown to affect the onset and subsequent development of the vortex flow field. Here we present results concerning suppression and control of the onset of instability as well as demonstrating competition between different vortex mechanisms. These effects modulate the interfacial mass transport and, hence, ionic current, through the interface and produce observable patterns. These results are of both fundamental and practical interest, with implications regarding early transitions from limiting to over-limiting currents and colloid-hydrodynamic interactions. The practical applications of such effects range from bio-molecular concentration, separation, and detection to micro-purification and on-chip electro-dialysis.

9:57AM M6.00010 Characterization of electrochemical response of a hybrid micro-nanochannel system using computational impedance spectroscopy (CIS)[1] , VISHAL NANDIGANA, NARAYAN ALURU, Department of Mechanical Science and Engineering, Beckman Institute for Advanced Science and Technology, University of Illinois at Urbana Champaign — Single molecule/particle sensing using micro/nanochannel integrated systems has attracted tremendous interest in recent years. The molecule in an aqueous ionic solution is translocated from the source microchannel towards the drain microchannel across a nanochannel under the influence of an external electric field. The translocated molecules are characterized from the electrical response of the system. In order to develop an efficient design for accurate characterization of single molecules, it is important to first understand the ion-transport dynamics in these integrated systems. To this end, we develop a computationally efficient area-averaged multi-ion transport model (AAM), considering an ion-selective nanochannel integrated with a microchannel on either side. Further, we study the ion transport dynamics both under equilibrium and non-equilibrium regimes. In each regime, the base state is perturbed with an external harmonic electrical disturbance over a wide range of frequency spectrum and the electrochemical impedance response is computed. We correlate each characteristic frequency present in the system to its corresponding physical phenomena and also characterize the microscopic diffusion boundary layer lengths (DBL) observed in the microchannel.

[1]This work was supported by the National Science Foundation (NSF) under Grants 0328162 (nano-CEMMS, UIUC), 0852657 and 0915718.

Tuesday, November 26, 2013 8:00AM - 10:10AM –

Session M7 Nanofluids II 329 - Reza Sadr, Texas A&M University at Qatar

8:00AM M7.00001 A Statistical Perspective on the Effects of Brownian Particle Movements on the Induced Fluid Flow Field , WAY LEE CHENG, REZA SADR, Texas A&M University — Nanofluids, engineered fluids by dispersing nanometer-sized materials in a base fluid, are reported to have anomalous heat transfer characteristics. In spite of the large number of, sometimes conflicting, reports on the existence and magnitude of enhancement, the underlying principles governing the improvements in the heat transfer process is not thoroughly understood. The interaction between the discrete and continuous phases in the fluid is thought to be a major contributor of the observed phenomena. The current study examines the fluid-particles interactions, induced by randomly moving particles suspended in a base fluid, from a statistical perspective, using an affordable computational approach. The fluid-particle interactions are described by Navier-Stokes equations on the fluid phase coupled with Langevine equation for the random walk of the suspended nano particles. Effects of particle diameter, unsteady movement, and hydrodynamics inertia in the fluid are examined in the current model. Results of the simulations show that the random movements of the particles induce a small random flow within the fluid. Statistics of the induced flow field converges asymptotically as the Brownian time-step reduces.

8:13AM M7.00002 Non-additive entrance effects in ionic conductance of an array of solid-state nanopores , ALESSANDRO GADALETA, CATHERINE SEMPERE, SIMON GRAVELLE, REMY FULCRAND, ALESSANDRO SIRIA, ILM, Université Lyon 1 and CNRS, UMR 5306, Villeurbanne, France, LYDÈRIC BOCQUET, ILM, Université Lyon 1 and CNRS, UMR 5306, Villeurbanne, France; MIT, Department of Chemical Engineering, Cambridge, MA — The ionic conductance of small pores has long been a topic of interest in many diverse areas of application, starting from electrophysiology in the 1950s to research on ultrarapid DNA sequencing and ion selective membranes in recent times. The so-called access resistance, induced by the convergence of field lines from the electrode to the pore, gives a significant contribution to the total ionic resistance. Here we investigate, experimentally and numerically, the access resistance of an array of solid-state nanopores, and show that it is not additive. We show how this problem can be approximately solved with simple electrostatics, presenting a matrix formalism which allows to quickly estimate the entrance effects in any given geometry.

8:26AM M7.00003 Rheological assessment of nanofluids at high pressure high temperature[1] , ANOOP KANJIRAKAT, REZA SADR, Texas A&M University at Qatar — High pressure high temperature (HPHT) fluids are commonly encountered in industry, for example in cooling and/or lubrications applications. Nanofluids, engineered suspensions of nano-sized particles dispersed in a base fluid, have shown prospective as industrial cooling fluids due to their enhanced rheological and heat transfer properties. Nanofluids can be potentially utilized in oil industry for drilling fluids and for high pressure water jet cooling/lubrication in machining. In present work rheological characteristics of oil based nanofluids are investigated at HPHT condition. Nanofluids used in this study are prepared by dispersing commercially available SiO_2 nanoparticles ($\sim$20nm) in a mineral oil. The basefluid and nanofluids with two concentrations, namely 1%, and 2%, by volume, are considered in this investigation. The rheological characteristics of base fluid and the nanofluids are measured using an industrial HPHT viscometer. Viscosity values of the nanofluids are measured at pressures of 100kPa to 42MPa and temperatures ranging from 25°C to 140°C. The viscosity values of both nanofluids as well as basefluid are observed to have increased with the increase in pressure.

[1]Funded by Qatar National Research Fund (NPRP 08-574-2-239).

8:39AM M7.00004 Automated Characterization and Sorting of Nanowires by Solution-Based Electro-Orientation Spectroscopy , CEVAT AKIN, JERRY SHAN, Rutgers University — The electrical conductivity and/or permittivity of nanowires and nanotubes are often poorly known and difficult to measure, requiring cleanroom-based microfabrication and precision positioning to measure directly. Traditional direct-characterization methods are also not compatible with further solution-based processing of nanowires. Electro-orientation spectroscopy, the rotation of nanowires in liquid suspension into alignment with external AC electric fields of different frequency, offers an alternative measurement technique that is simple and also compatible with further solution-based sorting and positioning of particles. We present the theory and our experimental results obtained by optical microscopy on the alignment rate of suspended nanowires of known conductivities under spatially uniform AC electric fields of different frequency. The deduced electrical conductivities of the nanowires are compared to direct 2-point-probe measurements. We demonstrate the compatibility of the electro-orientation method with further solution-based processing by implementing the technique in a novel microfluidic device capable of automated electrical characterization and sorting of nanowires.

8:52AM M7.00005 Collective alignment of nanorods in thin Newtonian films[1] , YU GU, RUSLAN BURTOVYY, JAMES TOWNSEND, Clemson University, JEFFERY OWENS, Air Force Reserach Laboratory, IGOR LUZINOV, KONSTANTIN KORNEV, Clemson University — We provide a complete analytical description of the alignment kinetics of magnetic nanorods in magnetic field. Nickel nanorods were formed by template electrochemical deposition in alumina membranes from a dispersion in a water–glycerol mixture. To ensure uniformity of the dispersion, the surface of the nickel nanorods was covered with polyvinylpyrrolidone (PVP). A 40–70 nm coating prevented aggregation of nanoroda. These modifications allowed us to control alignment of the nanorods in a magnetic field and test the proposed theory. An orientational distribution function of nanorods was introduced. We demonstrated that the 0.04% volume fraction of nanorods in the glycerol–water mixture behaves as a system of non-interacting particles. However, the kinetics of alignment of a nanorod assembly does not follow the predictions of the single-nanorod theory. The distribution function theory explains the kinetics of alignment of a nanorod assembly and shows the significance of the initial distribution of nanorods in the film. It can be used to develop an experimental protocol for controlled ordering of magnetic nanorods in thin films.

[1]This work was supported by the Air Force Office of Scientific Research, Grant numbers FA9550-12-1-0459 and FA8650-09-D-507 5900.

9:05AM M7.00006 Correlation between translational and rotational diffusion of a Janus nanoparticle in explicit solvent: A molecular dynamics simulation study , ALI KHARAZMI, Michigan State University, NIKOLAI PRIEZJEV, Wright State University — Molecular dynamics simulations are used to study the diffusion of a single Janus particle immersed in a Lennard-Jones fluid. We consider a spherical particle with two hemispheres of different wettability. We analyzed the time dependence of the orientation tensor, particle displacement, and translational and rotational velocity autocorrelation functions. It was found that both translational and rotational diffusion coefficients increase with decreasing surface energy of the nonwetting hemisphere. We also observed that in contrast to homogeneous particles, the nonwetting hemisphere of the Janus particle tends to rotate in the direction of the displacement vector during the rotational relaxation time.

9:18AM M7.00007 Molecular Dynamics of Reaction-Driven, Diffusiophoretic, Colloid Self-Propulsion , NIMA SHARIFI-MOOD, Levich Institute, Department of Chemical Engineering, City College of New York, JOEL KOPLIK[1], Department of Physics, City College of New York, CHARLES MALDARELLI, Levich Instiute, Department of Chemical Engineering, City College of New York — Chemical-mechanical transduction mechanisms which can actuate the movement of colloids through pathways in liquids are highly sought after as engines to propel miniaturized micro and nanobots. One mechanism involves harnessing van der Waals attractive forces between the colloid and solute molecules. Self propulsion can be achieved by arranging for the solute to react on one face of the colloid, creating an asymmetric distribution which can propel the particle. We use molecular dynamics calculations to elucidate this propulsion for nanocolloids. The calculations assume Lennard-Jones interactions between the colloid (modelled as a rigid cluster of atoms), solvent atoms and solute atoms which react with the colloid atoms on one face of the cluster. The solute reacts when localized within the attractive landscape of the cluster atoms and is converted for simplicity to solvent. Quantitative calculations of the diffusiophoretic velocity demonstrate the interplay of Brownian rotation and diffusiophoretic propulsion, the dependence of the nano-colloid velocity on its radius and an agreement with a continuum model which therefore allows a description of the phenomena for propulsion of objects in size and over trajectories from the nanometer to the micron scale.

[1]Levich Instiute, Department of Physics, City College of New York

9:31AM M7.00008 Repulsion parameters for carbon nanotubes in water in Dissipative Particle Dynamics simulations , MINH VO, DIMITRIOS PAPAVASSILIOU, The University of Oklahoma — In Dissipative Particle Dynamics (DPD) simulations, the thermodynamic and transport properties of the DPD fluid are governed by the selection of the repulsion, dissipation, and random noise parameters, because these parameters control the interaction potential and the motion of each DPD bead at each time step. For the case of the motion of carbon nanotubes in a water nanochannel, appropriate choices need to be made to ensure that DPD beads represent the system. The dissipative parameter (γ) should be equal to 4.5 for simulation stability. The noise parameter (σ) can be calculated using dissipation fluctuation relation, when the temperature of the system reaches equilibrium. In order to determine the repulsion parameter (a_{ij}) of CNT and water, we simulate the case of water flow past an array of single-walled CNTs. In this case, results from molecular dynamics simulations by Walther et al. (Phys. Rev. E, 2004, 062201) are available and can be used for validation. The hydrodynamic properties for a (32,0) single-walled CNT (32,0) with diameter of 2.5 nm were determined in different Reynolds number flows. With aij = 60 (kT/r_c), the drag coefficients of the CNT are quite similar to values from the analytical solution of the Stocks – Oseen equation. Additionally, the slip length on the CNT wall is comparable with the Walther et al. results. In addition, the application of these parameters in longer length scales and time scale will be discussed by increasing the number of water molecules grouped into each DPD bead.

9:44AM M7.00009 Fast Rotation of a Single Water Molecule in Buckyball , AMIR BARATI FARIMANI, YANBIN WU, NARAYANA ALURU, Mechanical Science and Engineering, Beckman Institute, University of Illinois at Urbana-Champaign — Successful encapsulation of a single water molecule in C60 by Kurotobi and Murata (Science, 333, 2011) opens up an opportunity to study non-hydrogen bonded single water molecule. Here, we investigate the properties of a single water molecule in buckyball by using molecular dynamics (MD) and density functional theory (DFT). By using DFT, we found that there's a shift of 23 cm^{-1} in vibrational frequencies of O-H bond of water molecule when it's inside C60. By using MD, we compute the rotational diffusion and entropy of water. Our findings show that water rotates about an order of magnitude faster compared to a single water molecule in bulk. While H_2O@C60 has near zero translational entropy, its rotational entropy is 6.5 times larger than rotational entropy of bulk water.

9:57AM M7.00010 Electrophoretic mobility of spherical nanoparticles confined in nanochannels , YU-WEI LIU, TOM WYNNE, SUMITA PENNATHUR, CARL MEINHART, UCSB — We investigate the mobility of a charged spherical nanoparticle driven by weak electric fields that are confined in nanochannels. Factors affecting mobility include particle zeta potential, electrolyte concentration, and channel size. Classic models for electrophoretic mobility (e.g. Smoluchowski and Huckel) are valid only in the linear regime of small particle zeta potential, and for an unbounded fluid domain. The classical models fail to predict electrophoretic mobility estimated from experiments using $\sim$ 42 nm diameter particles confined in a $\sim$ 100 nm nanochannel. We adopt the asymptotically-expanded formulations of Khair and Squires (Phys. Fluids, 2009), and solve the fully-coupled equations on a well-resolved 3D finite element domain. For a charged 42 nm diameter nanoparticle, confined in a 100 nm high nanochannel, the electrophoretic mobility increases nonlinearly when particle zeta potential is greater than thermal potential k_BT/e. When the channel size is decreased from 2.5 um to 100 nm, the mobility is reduced by up to 20%. The result suggests that particle/wall interactions, including overlapping double layers may affect electrophoretic mobility in a non-linear manner.

Tuesday, November 26, 2013 8:00AM - 10:10AM –

Session M8 General Fluid Dynamics II: Theory I 330 - Mark Hoefer, North Carolina State University

8:00AM M8.00001 Size-Dependent Fluid Mechanics , ALI HADJESFANDIARI, AREZOO HAJESFANDIARI, GARY DARGUSH, University at Buffalo, State University of New York — Classical fluid mechanics provides a reasonable basis for analyzing the behavior of fluid flow at the macro scale. However, experiments show that the behavior of fluid in small scales is different from their behavior at macro scales. An additional concern relates to the absence of a length scale in the governing Navier-Stokes equations, when the present description of turbulence seems to need the clear definition of a characteristic size. Consequently, there is need for a more complete fluid dynamics, which spans many scales and, of course, must reduce to classical fluid mechanics for flows with macro-scale size. Here we develop the consistent size-dependent fluid mechanics by discovering the skew-symmetric character of couple stress tensor. As a result, the skew-symmetric mean curvature rate vector as the consistent measure of deformation is introduced. It is demonstrated that this theory may provide a basis for fundamental studies of flows at the finest scales for which a continuum representation is valid and, perhaps, for gaining additional insight into the problem of turbulence.

8:13AM M8.00002 An Implicit Immersed Boundary Method for Low Reynolds Number Incompressible Flows[1] , HYUN WOOK PARK, CHANGHOON LEE, JUNG-IL CHOI, Dept. of CSE, Yonsei Univesity — We develop a new formulation of immersed boundary (IB) method based on direct forcing for incompressible viscous flows. The new algorithm for the present IB method is derived using a block LU decomposition and Taylor series expansion, and the direct forcing for imposing no-slip condition on the IB surface is calculated in an iterative procedure. We perform simulations of two-dimensional flows around a circular cylinder and three-dimensional flows over a sphere for low and moderate Reynolds numbers. The result shows that present method yield a better imposition of no-slip condition on IB surface for low Reynolds number with a fairly larger time step than other IB methods based on direct forcing.

[1]Supported by EDISON (2011-0029561) program of NRF

8:26AM M8.00003 Anisotropy in RT flows , YE ZHOU, W. CABOT, LLNL — This work investigates several key statistical measurements of turbulence induced by Rayleigh-Taylor instability using data from well resolved numerical simulations at moderate Reynolds number with the goal of determining the degree of departure of this inhomogeneous flow from that of homogeneous, isotropic turbulence. The simulations use two miscible fluids with unity Schmidt number and moderate density contrast (3/2 to 9). The results of this study should find application in subgrid-scale modeling for large-eddy simulations and Reynolds-averaged Navier-Stokes modeling used in many engineering and scientific problems.

8:39AM M8.00004 Interfacial dynamics of dissolving objects in fluid flow , CHRIS RYCROFT, University of California, Berkeley; Lawrence Berkeley National Laboratory; Harvard University, MARTIN BAZANT, Massachusetts Institute of Technology — An advection–diffusion-limited dissolution model of an object being eroded by a two-dimensional potential flow will be presented. By taking advantage of conformal invariance of the model, a numerical method will be introduced that tracks the evolution of the object boundary in terms of a time-dependent Laurent series. Simulations of several dissolving objects will be shown, all of which show collapse to a single point in finite time. The simulations reveal a surprising connection between the position of the collapse point and the initial Laurent coefficients, which was subsequently derived analytically using residue calculus.

8:52AM M8.00005 Low-dimensional modelling of high-Reynolds-number shear flows incorporating constraints from the Navier-Stokes equation , MACIEJ BALAJEWICZ, Stanford University, EARL DOWELL, Duke University, BERND NOACK, Institut PPRIME — We generalize the POD-based Galerkin method for post-transient flow data by incorporating Navier-Stokes equation constraints. In this method, the derived Galerkin expansion minimizes the residual like POD, but with the power balance equation for the resolved turbulent kinetic energy as an additional optimization constraint. Thus, the projection of the Navier-Stokes equation on to the expansion modes yields a Galerkin system that respects the power balance on the attractor. The resulting dynamical system requires no stabilizing eddy-viscosity term–contrary to other POD models of high-Reynolds-number flows. The proposed Galerkin method is illustrated with three test cases: two-dimensional flow past a stationary cylinder, two-dimensional flow inside a square lid-driven cavity and a two-dimensional mixing layer. Generalizations for more Navier-Stokes constraints, e.g. Reynolds equations, can be achieved in straightforward variation of the presented results.

9:05AM M8.00006 Stratified Euler flows in a channel and conservation laws , GIOVANNI ORTENZI, Università degli studi di Milano Bicocca, ROBERTO CAMASSA, SHENGQIAN CHEN, University of North Carolina, Chapel Hill, GREGORIO FALQUI, MARCO PEDRONI, Università degli studi di Milano Bicocca — We analyze the consequences of density stratification for the motion of an incompressible two dimensional Euler fluid confined to move under gravity between rigid lids and otherwise free to move along horizontal directions. The conserved quantity related to the horizontal translation invariance (impulse) does not coincide with the horizontal momentum, which is not conserved for generic initial conditions. The classical form of the impulse is given by Benjamin (1986) and it is affected by the boundary limiting values of physical fields such as density or density weighted vorticity. Therefore, the intersection between isopycnals and boundaries could affect the conservation laws of the system even if symmetries are not broken. While the failure to conserve quantities is naturally implied by geometrical changes of the fluid-domain boundary, regardless of fluid stratification, in this case the fluid domain maintains invariance under symmetry and the relevant cause of the failure is the connection properties of pycnoclines. Some results on this topological (non)conservation are exposed in the examples of impulse and total circulation.

9:18AM M8.00007 Shock Waves in Dispersive Eulerian Fluids[1] , MARK HOEFER, Mathematics, North Carolina State University — Shock waves in dispersive media with negligible dissipation are studied in the context of the compressible Euler equations with weak dispersion. Example fluids of this type include superfluids, shallow water flows, and ion-acoustic plasma. A characterization of one-dimensional dispersive shock waves (DSWs) will be presented. DSWs are sharply distinct from classical, dissipatively regularized shock waves both in terms of physical significance and mathematical description. Drawing on terminology from classical gas dynamics, jump conditions (shock loci and speeds) and admissibility criteria for the long time evolution of step-like initial data will be presented utilizing a nonlinear wave averaging technique. While entropy conditions determine admissible, dissipatively regularized shock waves, conservative, dispersive systems are time reversible and can exhibit positive or negative dispersion. The universal structure of weak shocks will be shown to depend solely upon the dispersion sign and pressure law. Large amplitude DSWs can exhibit novel effects such as cavitation and "implosion" yielding internal, multi-phase dynamics.

[1]Support from NSF DMS-1008973

9:31AM M8.00008 Wavelet-based Simulations of Unsteady Compressible Flows[1] , ERIC BROWN-DYMKOSKI, OLEG V. VASILYEV, University of Colorado Boulder — In this talk we present an extension of adaptive wavelet-based methodologies for unsteady compressible fluid simulations. This approach takes advantage of spatio-temporal intermittency of unsteady flows through a dynamically adaptive grid. It is built upon the adaptive wavelet collocation method, which allows for efficient mesh refinement at each time step with the error well-bounded by a prescribed threshold. Several benchmark simulations have been performed for compressible subsonic flows, including turbulent channel flow and flow around a bluff body. Atypically for channel flow simulations, a dyadic adaptive grid was used instead of the usual stretched mesh. While the external flow simulations are at a subcritical Reynolds number, spanwise instabilities create vortex loops that lead to a complex, three-dimensional wake. This work provides the basis for continuing development of adaptive, compressible turbulence models, including wavelet-based adaptive LES where the filter threshold is dynamically prescribed by global or local criteria.

[1]This work was supported by NSF under grant No. CBET-1236505.

9:44AM M8.00009 Non-uniqueness of solutions in asymptotically self-similar shock reflections , SEBASTIEN SM. LAU-CHAPDELAINE, MATEI I. RADULESCU, University of Ottawa — The present study numerically addresses the self-similarity of an unsteady shock reflection on an inclined wedge. The wedge-tip conditions are modified, allowing for a finite radius of curvature, and the following shock reflection configuration is observed at large distances from the tip. It is found that the type of shock reflection observed far from the corner, namely regular or Mach reflection, depends intimately on the wedge tip geometry, as the flow "remembers" how it was started. Substantial differences from a sharp-tipped wedge (without curvature) were found. For example, a shock with incident Mach number $M = 6.6$ and an isentropic exponent $\gamma = 1.2$ reflecting over wedge with a sharp tip will result in a Mach reflection when a wedge angle of 44° is used, while a 45° wedge will result in a regular reflection. This transition angle increases to between 57° and 58° when a wedge with a concave, curved tip is introduced. A vanishing length scale is introduced with the curved tip in a way which is similar to those of viscous and relaxation effects. While the length scale only dominates the solution at early times, this study shows that its effects play a dominant role in determining the asymptotic pseudo-steady shock reflection configuration.

9:57AM M8.00010 Single series skewness representation for passive scalar advection in laminar pipe and channel flow[1] , RICHARD M. MCLAUGHLIN, FRANCESCA BERNARDI, ROBERTO CAMASSA, University of North Carolina, KEITH MERTENS, Leap Motion, UNC JOINT FLUIDS LAB TEAM — In this talk, we present an exact single series representation for scalar skewness time evolution. Prior studies have naturally derived multiple nested Fourier series solutions which suffer from slow convergence and cloud physical interpretation. Judicious change of variables and complex residue theory lead to single series representation formulae for the moments along streamwise slices from which quantities such as variance and skewness can be reconstructed. Instantaneous symmetry breaking gives rise to non-zero skewness on transient time scales arising as a non-trivial competition between advection and diffusion which is captured by the simplified formulae. Small and long time asymptotics will be discussed for the first three moments in both channel and pipe geometries in steady Poiseuille flow, and nontrivial Peclet dependence in the skewness along slices will be examined.

[1]DMS-0502266, NSF RTG DMS-0943851, NSF RAPID CBET-1045653, NSF CMG ARC- 1025523, and NSF DMS-1009750, ONR DURIP N00014-09-1-0840

Tuesday, November 26, 2013 8:00AM - 9:31AM –

Session M9 Instability: Interfacial and Thin-Film VI - Fingering 333 - Ya Liu, University of Pittsburgh

8:00AM M9.00001 Stability Results on Multi-Layer Hele-Shaw Flows , CRAIG GIN, PRABIR DARIPA, Texas A&M University — Saffman-Taylor instability, which occurs when a less viscous fluid drives a more viscous fluid, has been studied for many years and has a wide range of applications. In particular, an understanding of this phenomenon is helpful in the attempt to maximize the effectiveness of chemically enhanced oil recovery techniques. We study this instability through linear stability analysis of multi-layer radial Hele-Shaw flows of immiscible fluids. We take classic results on the instability of flows consisting of two fluids and extend them to flows with an arbitrary number of fluid phases. Using upper bound results on the growth rate of instabilities obtained in this general setting, we are able to give conditions under which this regime is less unstable than the single interface case.

8:13AM M9.00002 A Solutal Fingering Instability during Capillary Imbibition in Porous Media , CHRISTOPHER GUIDO, NICHOLAS YOUNG, WILLIAM RISTENPART, Dept. Chemical Engineering & Materials Science, University of California, Davis — We report the existence of a solute-driven fingering instability that occurs during capillary imbibition into cellulosic porous media. Contacting a piece of paper with an aqueous solution containing hydrophobic solutes causes the liquid to move forward into the paper. For sufficiently low solute concentrations and sufficiently high ambient humidities, the imbibition front moves forward smoothly as expected. For higher concentrations and lower humidities, however, the imbibition front develops spatially periodic oscillations that grow with time, i.e., a fingering instability occurs. Surprisingly, under these conditions the solute concentration becomes larger at the imbibition front compared to the bulk, contrary to the behaviour expected based on chromatographic separation. We present a stability analysis predicated on solutal changes in the interfacial tension driven by water imbibition into a precursor film ahead of the macroscopically observable air/water interface, and we derive a critical Péclet number above which the interface is unstable.

8:26AM M9.00003 Magnetically induced solitons in a Hele-Shaw cell[1] , SERGIO LIRA, JOSE MIRANDA, Universidade Federal de Pernambuco, Recife, Brazil — We study the development of propagating solitons on the interface separating two viscous fluids flowing in parallel in a vertical Hele-Shaw cell. One of the fluids is a ferrofluid and a uniform magnetic field is applied in the plane of the cell, making an angle with the initially undisturbed interface. We derive a Korteweg-de Vries equation for long waves at this confined geometry which predicts the possibility of controlling the speed of the solitons by magnetic means. The influence of the tilted magnetic field on the velocity and on the shape of the solitary waves is investigated.

[1]We thank CNPq (Brazilian Research Council) for financial support.

8:39AM M9.00004 Effect of transient interfacial tension on miscible viscous fingering[1] , MANORANJAN MISHRA, SATYAJIT PRAMANIK, Indian Institute of Ropar, India — The pressure-driven displacement flow of a more viscous fluid by a less viscous one is an unstable configuration in the context of miscible viscous fingering in porous media. Steep concentration, density or temperature gradient at the interface of the underlying fluids gives rise to a nonconventional stress in the system, which causes an effective or transient interfacial tension. Such tension has been incorporated using Korteweg stresses in the momentum equation. The system has been modeled by coupling the continuity and Darcy-Korteweg equations with the convection-diffusion equation for the evolution of the solvent concentration. We have shown by a numerical simulation based on Fourier-spectral method that such system can remain stable for a comparatively longer initial transient period. This delay on the onset of instability is due to transient interfacial tension acting at the miscible diffusive interface. The results show that increasing the strength of the stress the onset of instability can be delayed significantly. However, the system may or may not become completely stable depending upon the configuration of the displaced fluid. Linear stability analysis of such system having two semi-infinite fluids and with a finite slice of fluid has been also investigated.

[1]Financial support from DST and NBHM Government of India is gratefully acknowledged.

8:52AM M9.00005 Strong sample solvent and viscous fingering effects on the dynamics of adsorbed solutes , CHINAR RANA, Indian Institute of Technology Ropar, India, ANNE DE WIT, Université Libre de Bruxelles, Belgium, MICHEL MARTIN, PMMH-ESPCI, Paris, France, MANORANJAN MISHRA, Indian Institute of Technology Ropar, India — The pressure driven displacement flow in a porous medium with viscosity increasing in the direction of the flow leads to viscous fingering of the rear interface of finite samples. Sample solvent effects exist if the adsorption constant of solutes on the porous matrix depends on the solvent composition. A sample solvent stronger than the displacing fluid then leads to spatially variable retention of the solute initially dissolved in the sample. We investigate here the influence of these two effects, variable retention and viscosity contrast, on the dynamics of the solute. The continuity equation and Darcy's law coupled to convection-diffusion equations for the evolution of the sample and solute concentration are solved numerically to analyze the above phenomena. The sample viscosity and solute retention are assumed to depend exponentially on the concentration of a solute initially contained in the sample. The results demonstrate the development of two solute concentration zones, one of them being affected by the viscous fingering pattern. The effect of the fingering instability on the retained solute zone increases with an increase in the strength of the sample solvent. This, in turn, increases the spreading zone of the solute and delays the disengagement of the solute from the sample zone.

9:05AM M9.00006 Wavelength selection in injection-driven Hele-Shaw flows: A maximum amplitude criterion[1] , EDUARDO DIAS, JOSE MIRANDA, Depto de Fisica - Univ Federal de Pernambuco — As in most interfacial flow problems, the standard theoretical procedure to establish wavelength selection in the viscous fingering instability is to maximize the linear growth rate. However, there are important discrepancies between previous theoretical predictions and existing experimental data [T. Maxworthy, Phys. Rev. A **39**, 5863 (1989)]. In this work we perform a linear stability analysis of the radial Hele-Shaw flow system that takes into account the combined action of viscous normal stresses and wetting effects. Most importantly, we introduce an alternative selection criterion for which the selected wavelength is determined by the maximum of the interfacial perturbation amplitude. The effectiveness of such a criterion is substantiated by the significantly improved agreement between theory and experiments.

[1]We thank CNPq (Brazilian Sponsor) for financial support

9:18AM M9.00007 Determining the number of fingers in the lifting Hele-Shaw problem[1] , JOSE MIRANDA, EDUARDO DIAS, Depto de Fisica - Univ Federal de Pernambuco — The lifting Hele-Shaw cell flow is a variation of the celebrated radial viscous fingering problem for which the upper cell plate is lifted uniformly at a specified rate. This procedure causes the formation of intricate interfacial patterns. Most theoretical studies determine the total number of emerging fingers by maximizing the linear growth rate, but this generates discrepancies between theory and experiments. In this work, we tackle the number of fingers selection problem in the lifting Hele-Shaw cell by employing the recently proposed maximum-amplitude criterion [Dias and Miranda, Phys. Rev. E **88**, 013016 (2013)]. Our linear stability analysis accounts for the action of capillary, viscous normal stresses, and wetting effects, as well as the cell confinement. The comparison of our results with very precise laboratory measurements for the total number of fingers shows a significantly improved agreement between theoretical predictions and experimental data.

[1]We thank CNPq (Brazilian Sponsor) for financial support

Tuesday, November 26, 2013 8:00AM - 10:10AM –

Session M10 Instability: General III - Stratified and Planar Flow, Cavity Flow and Periodic Orbits 334 - Rama Govindarajan, Tata Institute of Fundamental Research

8:00AM M10.00001 Self-Sustained Oscillations of Flow Past Sequential Cavities: Effects of Gravity Wave Coupling , BURAK A. TUNA, DONALD ROCKWELL, Lehigh University — Shallow flow past successive cavities can lead to highly coherent oscillations, due to coupling between: the inherent instability of the separated shear layer along the opening of each cavity; and a gravity standing wave mode within the cavity. As the flow velocity is varied, this coupling is associated with different orientations of the gravity standing wave, i.e., it can occur in either the transverse or the streamwise direction. The flow structure along the separated shear layer and within the cavity is, in turn, a strong function of the orientation and phase of the standing wave. When the oscillation amplitude of the coupled instability- cavity mode becomes large, as indicated by the amplitude of deflection of the free-surface, enhanced coherence and scale of the phase-averaged vortex formation occurs in the separated shear layer along the opening of the cavity. This coherent vortex formation results in a large increase in the magnitude of the turbulent shear stresses in the separated shear layer and, as a consequence, an increase of the time-averaged exchange velocity and mass exchange coefficient along the opening of the cavity. Furthermore, the flow structure and mass exchange along each of the sequential cavities may be either substantially different or very similar, depending on the orientation and phase of the gravity standing wave within the cavity, that is, a streamwise-oriented versus a transverse-oriented gravity standing wave, as well as the phase shift of the oscillations occurring in adjacent (sequential) cavities.

8:13AM M10.00002 Fast and slow transition to turbulence in plane Poiseuille flow , BRUNO ECKHARDT, STEFAN ZAMMERT, Philipps-Universitat Marburg — Plane Poiseuille flow has two paths to turbulence: a slow one connected with a linear instability to Tollmien-Schlichting waves at Reynolds numbers above 5772, and a fast one through a by-pass transition at much lower Reynolds numbers. We explore the conditions for the two transition scenarios and their connections in the state space of the system by tracking the time evolution of different perturbations, i.e. we use the edge tracking algorithm for the identification of edge states (PRL 96, 174101 (2006)). We identify the two travelling waves that govern the transition process and study their subcritical bifurcations. The fast transition is realized for a large set of initial conditions as soon as it appears. The slow transition process first appears in a very thin slice that grows with Reynolds number but becomes noticeable only shortly before the linear instability. Both transition paths are shown to converge to the same turbulent state.

8:26AM M10.00003 Stability Analysis of High-Speed Cavity Flow[1] , YIYANG SUN, KUNIHIKO TAIRA, LOUIS CATTAFESTA, Florida State University, GUILLAUME BRES, Cascade Technologies, LAWRENCE UKEILEY, University of Florida — Stability analysis is conducted to uncover the inherent instabilities in subsonic to supersonic open cavity flows. Two- and three-dimensional direct numerical simulations of spanwise periodic cavity flows are performed with the high-fidelity compressible flow solver "Charles" developed at Cascade Technologies. Two-dimensional nonlinear computations are carried out to characterize the flow stability over a wide range of Mach numbers and Reynolds numbers, and to extract a base flow for three-dimensional linear stability analysis. Selective frequency damping method is used to solve for the steady state for cases where the flow is found to be unstable. Both stable and unstable two-dimensional steady state can then be used as base state to examine, in the linear limit, how instabilities grow in space and over time. The present study forms a foundation to pursue three-dimensional flow control in which the spanwise instability will be exploited to redistribute kinetic energy from large spanwise vortices to reduce load fluctuations within the cavity.

[1]Work supported by AFOSR (grant FA9550-13-1-0091).

8:39AM M10.00004 Unstable periodic orbits in a homogeneous shear flow[1] , ATSUSHI SEKIMOTO, SIWEI DONG, JAVIER JIMÉNEZ, U. Politécnica Madrid — Unstable periodic orbits (UPOs) are numerically obtained by a Newton-Krylov method in a homogeneous shear flow. The two classes of UPOs have a box-time period synchronized with that of the boundary condition, which is shear-periodic between shifting points of the upper and bottom boundaries of the computational box. The first one is characterized by the shift-reflection symmetry, and by staggered streamwise-inclined vortex pairs, as in Nagata's Couette equilibrium solution (JFM 217, 519-527 (1990)). The second is characterized by the mirror symmetry in the spanwise direction, similar to Townsend's sketch of a inclined double-roller eddy (JFM 41, 13-46 (1970)). It is revealed that the lower branch of the mirror-symmetric UPO has an important role in the transition to turbulence, and is an "edge-state" on the basin boundary between laminar and turbulent states, whose two unstable directions lead to direct laminarization and to turbulence bursting. We also present subharmonic UPOs, whose periods are longer than the periodicity of the boundary condition. The dynamic UPO represents the breakdown and regeneration of streaks, associating with the streamwise-inclined vortices, which is similar to the self-sustaining process in wall-bounded flows.

[1]Funded by the ERC Multiflow project.

8:52AM M10.00005 Experimental Investigation of Fluid-Structure Interactions in Compressible Cavity Flows , JUSTIN WAGNER, KATYA CASPER, STEVEN BERESH, PATRICK HUNTER, RUSSELL SPILLERS, JOHN HENFLING, RANDALL MAYES, Sandia National Laboratories, SANDIA NATIONAL LABORATORIES TEAM — Experiments were performed to understand the complex fluid-structure interactions that occur during internal store carriage. A cylindrical store was installed in a cavity having a length-to-depth ratio of 3.33 and a length-to-width ratio of 1. The Mach number ranged from 0.6 – 2.5 and the incoming turbulent boundary layer thickness was about 30-40% of the cavity depth. Fast-response pressure measurements provided aeroacoustic loading in the cavity, while triaxial accelerometers and laser Doppler vibrometry provided simultaneous store response. Despite occupying only 6% of the cavity volume, the store significantly altered the cavity acoustics. The store responded to the cavity flow at its natural structural frequencies, as previously determined with modal hammer tests, and it exhibited a directional dependence to cavity resonance. Specifically, cavity tones excited the store in the streamwise and wall-normal directions consistently, while a spanwise response was observed only occasionally. The streamwise and wall-normal responses were attributed to the known pressure gradients in these directions. Furthermore, spanwise vibrations were greater at the downstream end of the cavity, attributable to decreased levels of flow coherence near the aft-wall.

9:05AM M10.00006 The genesis of streamwise-localized solutions from globally periodic travelling waves in pipe flow , MATTHEW CHANTRY, University of Bristol, ASHLEY WILLIS, University of Sheffield, RICH KERSWELL, University of Bristol — At intermediate Reynolds numbers, pipe flow exhibits spatio-temporal turbulence, where localized patches of turbulence may spread, split or decay. To construct a dynamical systems framework for this behaviour requires streamwise-localized solutions embedded within the turbulent dynamics. To date a single localized periodic orbit has been found, in contrast to the large number of known downstream-periodic solutions. Here we find the origin of this localized solution in a symmetry-breaking Hopf bifurcation from a known downstream periodic travelling wave. This bifurcation structure is found in a second symmetry subspace leading to new localized solutions. Our results indicate that localized versions of every downstream-periodic travelling wave should be expected.

9:18AM M10.00007 Transient growth of 3D perturbations in a stratified mixing layer flow , HELENA VITOSHKIN, Tel-Aviv University — Three-dimensional non-modal disturbances growth in a stably stratified viscous mixing layer flow is studies. The research is performed in the framework of linearized equations using two indepndent approaches and then is verified by computational modeling of evolution of the optimal perturbations found via numerical solution of fully non-linear time-dependent Boussinesq equations. We examined the effect of stratification on linearly stable three-dimensional disturbance, which attains the largest non-modal amplification in the non-isothermal case. The transient strong amplification could be reached at short times by a 3D optimal perturbation, whose amplitude grows larger than those computed in the 2D case, even in cases of very strong stable stratification. This non-modal growth is governed mainly by the Holmboe modes, and does not necessarily weaken with increase of the Richardson number.

9:31AM M10.00008 Linear optimal perturbations of a stratified shear layer , ALEXIS KAMINSKI, JOHN TAYLOR, DAMTP, University of Cambridge — Stratified shear flows are ubiquitous features of the ocean and atmosphere, and a large literature is devoted to describing their stability and mixing properties. A classical example is the Kelvin-Helmholtz instability that is possible when the gradient Richardson number is less than $1/4$ somewhere in the flow. Here, we use numerical simulations to seek the three-dimensional "optimal perturbations" which maximize the growth of perturbation energy over a finite time interval T. In the limit of a long time interval, we expect to recover the fastest growing linear normal mode. However, for shorter time intervals, enhanced transient growth is possible due to the non-normality of the governing equations. The Reynolds, Richardson, and Prandtl numbers of the flow are varied in our analysis, and the resulting optimal perturbations compared to predictions from theory and past work. Enhanced growth rates are observed for short times in both the linearly stable (i.e. $\mathrm{Ri} > 1/4$) and unstable cases, and the optimal perturbations found have more structure than the most unstable mode predicted by normal-mode stability theory.

9:44AM M10.00009 Instability in viscosity-stratified free shear layer , KIRTI SAHU, Department of Chemical Engineering, Indian Institute of Technology Hyderabad, Yeddumailaram 502 205, India, RAMA GOVINDARAJAN, TIFR Centre for Interdisciplinary Sciences, Tata Institute of Fundamental Research Narsingi, Hyderabad 500075, India — The stability of a mixing layer made up of two miscible fluids, with a viscosity-stratified layer between them, is studied. The two fluids are of the same density. It is shown that unlike other viscosity stratified shear flows, where species diffusivity is a dominant factor determining stability, species diffusivity variations over orders of magnitude do not change the answer to any noticeable degree in this case. Viscosity stratification, however, does matter, and can stabilize or destabilize the flow, depending on whether the layer of varying velocity is located within the less or more viscous fluid. This flow is a thus a prototype for a situation where viscosity stratification acts on the stability by an inviscid mechanism. This is confirmed by making an inviscid model flow with a slope change across the "viscosity" interface. The absolute instability of the flow can also be significantly altered by viscosity stratification.

9:57AM M10.00010 Absolute instability in viscoelastic mixing layers , PRASUN RAY, TAMER A. ZAKI, Imperial College London — The linear stability of viscoelastic planar mixing layers is investigated. The influence of viscoelasticity in dilute polymer solutions is modeled with the Oldroyd-B and FENE-P constitutive equations, and we examine how flow and viscoelastic parameters influence the onset of local absolute instability. With the Oldroyd-B model, the influence of the polymer is destabilizing, and this effect is almost fully captured by an elasticity parameter. Results obtained with the FENE-P model exhibit a rich variety of behavior. At large values of the maximum polymer extensibility, L, results are similar to those for the Oldroyd-B fluid as expected. However, when L is reduced to more realistic values, one must consider the ratio We/L (where We is the Weissenberg number), in addition to the elasticity. When We/L is large, the base-state polymer stress obtained by the FENE-P model is reduced relative to the Oldroyd-B stress. As a result, the overall influence of viscoelasticity on stability is reduced. Additionally, elasticity exhibits a stabilizing effect. As We/L is reduced, the FENE-P base-state polymer stress increases towards the Oldroyd-B value, and the destabilizing influence of elasticity observed with the Oldroyd-B model is again present.

Tuesday, November 26, 2013 8:00AM - 9:44AM –

Session M11 Non-Newtonian Flows II 335 - Sarah Kieweg, University of Kansas

8:00AM M11.00001 Electrokinetic particle motions in non-Newtonian fluids through a microchannel contraction , XINYU LU, Clemson University, SANG-WOO JOO, Yeungnam University, SHIZHI QIAN, Old Dominion University, XIANGCHUN XUAN, Clemson University — Electrokinetic flow is a powerful means to transport and control fluids and particles in microfluidic devices. To date, however, nearly all previous studies have used aqueous buffer solutions that are Newtonian fluids. We present our recent experimental results of the electrokinetic particle motions in a phosphate buffer with (which is a non-Newtonian fluid) and without (which is a Newtonian fluid) the addition of polyethylene oxide (PEO) through a planar contraction-expansion microchannel. We find that the viscoelasticity of the PEO solution can cause a particle bouncing phenomenon in the contraction, which has never been observed in Newtonian fluids. The effects of electric field, particle size, PEO concentration, and buffer concentration on this electrokinetic particle instability are experimentally examined.

8:13AM M11.00002 Reversibility and Chaos in Microscopic Fluid Systems , LIAT ROSENFELD, LIN FAN, SINDY K.Y. TANG, Stanford University, TANG RESEARCH GROUP TEAM — In this study, we explore the transition from reversible to chaotic behavior in an oscillatory shear flow of water-in-oil emulsions. Emulsions are complex materials and have many applications in chemical, biological and industrial processes. The many-body, shear-history-dependent nature of the microstructure renders the prediction of the complex material's dynamics and rheology highly nontrivial. The emulsion was injected through a microchannel and was forced to rearrange due to a central constriction in the channel. We study the motion of the individual droplets and their neighbors in order to determine their ability to retain their original position after several cycles of oscillations. We have found that while at the Stokes flow limit, the emulsion exhibit behaviors that vary from complete reversibility to complete irreversibility depending on the volume fraction, velocity and strain rate. We provide the first direct visualization of this phenomenon. This work is an important step in understanding the microscopic rearrangements of droplets and particles near jamming.

8:26AM M11.00003 A Computational Study of Viscoelastic Effects on Drop Dynamics in Microchannels[1] , DAULET IZBASSAROV, METIN MURADOGLU, Department of Mechanical Engineering, Koc University Rumelifeneri Yolu, Sariyer 34450 Istanbul, Turkey — A front-tracking method is developed and applied to study effects of viscoelasticity on drop dynamics in microchannels. The FENE-CR and Oldroyd-B models are employed to model the viscoelasticity. The viscoelastic model equations are solved fully coupled with the flow equations. An explicit semi-analytical time integration scheme is used for the viscoelastic model equations at low Deborah numbers and a log-conformation is used to alleviate the well-known difficulties at high Deborah numbers. The log-conformation is found to be stable and very robust for a wide range of Deborah numbers. The method is first validated for the benchmark single-phase viscoelastic flow through an axisymmetric channel with a 4:1 constriction and the results are found to be in a good agreement with earlier computational simulations. The algorithm is then used to study fluid dynamics of buoyancy-driven viscoelastic two-phase systems in a capillary tube. Extensive computations are performed to examine the effects of confinement and rheological properties of the phases on drop mobility and deformation. Finally, the method is applied to study the motion and deformation of a viscoelastic droplet in a pressure driven axisymmetric contraction/expansion micro-channel. Key Words:Viscoelastic fluid,FENE model.

[1]This work is supported by the Scientific and Technical Research Council of Turkey (TUBITAK), Grant No. 112M181.

8:39AM M11.00004 Falling Film Flow of Slag , LING MIAO, WEI-TAO WU, Department of Mechanical Engineering, Carnegie Mellon University, NADINE AUBRY, Department of Mechanical Engineering, Northeastern University, MEHRDAD MASSOUDI, U.S. Department of Energy, National Energy Technology Laboratory — In this paper, numerical calculations have been performed to study the heat transfer in the fully developed flow of a slag layer down a vertical wall. A new constitutive relation for the stress tensor of the slag is proposed, where the viscosity depends on the volume fraction, temperature, and shear rate. For the heat flux vector, we assume the Fourier's law of conduction with a constant thermal conductivity. The model is also capable of exhibiting normal-stress effects. The effects of various dimensionless numbers on the velocity, temperature and volume fraction are examined by numerically solving the governing equations. We also compared the different cases of shear thinning and shear thickening, cooling and heating. The effect of the exponent in the Reynolds viscosity model is also discussed. The results indicate that the viscous dissipation and radiation (at the free surface) cause the temperature to be higher inside the flow domain.

8:52AM M11.00005 Modeling flow of nematic liquid crystal down an incline[1] , MICHAEL LAM, LINDA CUMMINGS, New Jersey Institute of Technology, USA, TE-SHENG LIN, Loughborough University, UK, LOU KONDIC, New Jersey Institute of Technology, USA — The flow of nematic liquid crystals (NLCs) down an inclined substrate is studied. Under the usual long wave approximation, a 4th order nonlinear parabolic partial differential equation (PDE) of diffusion type is derived for the free surface height, $z = h(x, y, t)$. The model accounts for elastic distortions of the director field due to different anchoring conditions at the substrate and the free surface. The PDE we derive admits 2D traveling-wave solutions, which may translate stably or exhibit instabilities in the flat film behind the traveling front. These instabilities, which are distinct from the usual transverse instability of downslope flow, may be analyzed and explained by linear stability analysis of a flat translating film. Intriguing parallels are found with the instabilities exhibited by Newtonian fluid flowing on an inverted substrate [T-S. Lin and L. Kondic, Phys. Fluids **22**, 052105 1-10, (2010)].

[1]Supported by NSF grant DMS-1211713

9:05AM M11.00006 Influence of yield stress and shear thinning on the capillary ridge formation of gravity-driven Herschel-Bulkley fluid on an incline[1] , MD. RAJIB ANWAR, BIN HU, KYLE CAMARDA, SARAH KIEWEG, University of Kansas — In this work on gravity-driven spreading, we discuss the impact of surface tension on the spreading and free surface shape of a finite bolus of a Herschel-Bulkley fluid. We incorporate surface tension into a 2D (i.e. 1D spreading) Herschel-Bulkley thin film flow model. Studies have indicated that incorporating surface tension can result in the emergence of a capillary ridge in thin fluid films and the capillary ridge is strongly related to contact line fingering instability. Our previous numerical study showed that increased shear-thinning (in a fluid without yield stress) suppressed the capillary ridge. A previous linear stability analysis by Balmforth *et al.* (2007) showed that the yield stress in a Bingham fluid dampens the instability. Our numerical results in this study will provide initial insight on the impact of yield strength, shear-thinning index, and inclination angle on the overall spreading and appearance of the capillary ridge in Herschel-Bulkley fluids.

[1]This work is supported by NIH R21/R33 Microbicide Innovation Program IV [R21/R33 AI082697]

9:18AM M11.00007 The effect of the polymer relaxation time on the nonlinear energy cascade and dissipation of statistically steady and decaying homogeneous isotropic turbulence[1] , PEDRO C. VALENTE, CARLOS B. DA SILVA, IST/Technical University of Lisbon, Mecânica I, 1o andar/LASEF, Av. Rovisco Pais, 1049-001 Lisbon, Portugal, FERNANDO T. PINHO, Faculdade de Engenharia da Universidade do Porto (FEUP), Rua Dr. Roberto Frias, s/n, 4200-465 Porto, Portugal — We report a numerical study of statistically steady and decaying turbulence of FENE-P fluids for varying polymer relaxation times ranging from the Kolmogorov dissipation time-scale to the eddy turnover time. The total turbulent kinetic energy dissipation is shown to increase with the polymer relaxation time in both steady and decaying turbulence, implying a "drag increase." If the total power input in the statistically steady case is kept equal in the Newtonian and the viscoelastic simulations the increase in the turbulence-polymer energy transfer naturally lead to the previously reported depletion of the Newtonian, but not the overall, kinetic energy dissipation. The modifications to the nonlinear energy cascade with varying Deborah/Weissenberg numbers are quantified and their origins investigated.

[1]The authors acknowledge the financial support from Fundação para a Ciência e a Tecnologia under grant PTDC/EME-MFE/113589/2009

9:31AM M11.00008 Chemically-reacting non-linear fluid with variable transport properties , KEREM UGUZ, Bogazici University, Department of Chemical Engineering, MEHRDAD MASSOUDI, (2) U.S. Department of Energy, National Energy Technology Laboratory (NETL) — We study the momentum and the heat transfer of a chemically reacting non-linear fluid between two long horizontal plates which are kept at constant but different temperatures. The top plate is sheared at constant speed, while the bottom plate is kept stationary. The physical parameters of the fluid, i.e. the viscosity, the thermal conductivity and the diffusion coefficient are assumed to be a function of the concentration. The boundary value problem is solved numerically using Chebyshev Spectral Method. A detailed parametric study of the velocity, the temperature, and the concentration profiles are presented for shear-thinning/thickening and chemically-thinning/thickening fluids.

Tuesday, November 26, 2013 8:00AM - 9:57AM –

Session M12 Vortex Dynamics and Vortex Flows IX 336 - Kamran Mohseni, University of Florida

8:00AM M12.00001 On relation between scalar interfaces and vorticity in inviscid flows , O.N. RAMESH, SAURABH PATWARDHAN, Indian Institute of Science Bangalore — A great variety of applications like pollutant mixing in the atmosphere, mixing of reactants in combustion highlight the importance of passive scalar dynamics in fluid flows. The other dynamically important variable in the study of fluid flow is the vorticity. Vorticity though, unlike a passive scalar, does affect the fluid motion. The dynamics of scalar (linear) and vorticity (non-linear) are governed by the equations which inherently have different characteristics. This paper addresses the question of the faithfulness of representation of vorticity by scalar marker and the motivation for this comes from the experiment of Head and Bandyopadhyay (1981) which showed the existence of coherent vortices by using smoke flow visualization in a turbulent boundary layer. We will show analytically in regions where the molecular diffusion effects are negligible, the vorticity and scalar gradients are orthogonal to each other. The iso- surface of scalar follows the vorticity in an inviscid situation. Also, we will demonstrate that in the case of unsteady burgers vortex and vortex shedding behind a finite circular cylinder, the scalar gradient is orthogonal to vorticity and inner product of vorticity and scalar gradients is zero in regions away from the wall.

8:13AM M12.00002 Inviscid Damping of Vortex Asymmetries by a Critical Layer Flux[1] , C.F. DRISCOLL, A.A. KABANTSEV, C.Y. CHIM, T.M. O'NEIL, UCSD — Experiments and theory characterize a novel regime of near-inviscid 2D vortex symmetrization, wherein a weak flux through the critical layer causes algebraic (rather than exponential) damping of azimuthal asymmetries. This is distinct from exponential critical-layer damping (or spiral wind-up), where the damping may cease once the critical-layer vorticity is trapped in cats-eyes.[2] Here, weak viscosity causes slow vortex expansion and negligible direct azimuthal-shear damping; but when the weak expansion flux reaches the critical layer, previously un-damped Kelvin waves are rapidly damped to zero. Pure electron plasma experiments have quantitatively characterized this novel damping for $m_\theta = 1$ and $m_\theta = 2$ waves, obtaining wave amplitudes varying as $A(t) = A_0 - \gamma\, t$. A simple analysis of critical-layer dynamics agrees well with experiments for $m_\theta = 1$ waves (with a bounding wall); but suggests a $\gamma\, t^{2/3}$ dependence for $m_\theta = 2$ due to the critical-layer width scaling with wave amplitude. Simulations suggest that weak diffusion may obviate this discordant time exponent of 2/3.

[1]Supported by PHY-0903877 and DE-SC0002451.

[2]D.A. Schecter et al., Phys. Fluids **12**, 2397 (2000).

8:26AM M12.00003 Point vortex modeling of symmetric four vortex wakes , SAIKAT BASU, MARK STREMLER, Virginia Tech — Bluff body wakes frequently display different complex patterns. Our previously disseminated results introduced a point vortex model for the 2P wake configuration, which is one of the most commonly observed wake patterns and consists of two staggered rows of vortex pairs. In this talk, we present our findings for the related case of a symmetric 2P-like wake configuration. The pattern consists of two pairs of counter-rotating vortices placed symmetrically about the wake centerline in a singly-periodic domain. Due to the assumed symmetry of the wake we are able to model the vortex dynamics as an integrable Hamiltonian system. The mathematical analysis reveals some interesting and novel relative vortex motions that we will discuss. The model results will be compared against experimental wakes from the literature. As with our staggered 2P wake analysis, the model results suggest that the classification of these exotic wakes should include more than just the number of vortices shed by the body.

8:39AM M12.00004 Topological Classification of Periodic Solutions to the Point Vortex Model , SPENCER SMITH, Mount Holyoke College — The point vortex model represents one of the earliest attempts to discretize the field equations of fluid motion. Since the time of Helmholz, it has served as a starting point for the investigation of such disparate phenomena as weak turbulence and negative temperature states. Using this simple and elegant model, we created a large data set of numerically generated periodic orbits for small numbers of identical vortices. We then applied a topological classification scheme based on braid theory to organize and sort the data. This novel approach reveals unexpected and intriguing patterns in the distribution of these solutions in phase and parameter space.

8:52AM M12.00005 Desingularized propagating vortex equilibria[1] , STEFAN LLEWELLYN SMITH, MAE, UCSD — The correction to the propagation velocity of point vortex equilibria is calculated by allowing the vortices to have finite core size. A matched asymptotic expansion in the small parameter ϵ, given by the ratio of the core size to the dimension of the configuration, is carried out. The resulting velocity correction is found to be of order ϵ^4 and comes from the interaction of terms in the inner expansion. The results are compared to the known cases of propagating hollow vortex and vortex patch dipoles.

[1]This research was supported by National Science Foundation grant CMMI-0970113.

9:05AM M12.00006 Extreme Vortex States and the Growth of Palinstrophy in Two Dimensions , DIEGO AYALA, BARTOSZ PROTAS, McMaster University — We probe the sharpness of analytic estimates for the instantaneous rate of growth and the finite-time growth of palinstrophy in 2D viscous incompressible flows on periodic domains. This effort is part of a broader research program concerning a systematic search for extreme vortex states which is intrinsically related to the finite-time "blow-up" problem in 3D incompressible flows. Evidence is presented for the existence of a family of 2D vorticity fields parametrized by their energy and palinstrophy which saturate an estimate characterizing the finite-time growth of palinstrophy. The family of such "optimal" vortex states is obtained by solving suitable optimization problems in which the rate of growth of palinstrophy is maximized under constraints. Although found as a solution of an instantaneous problem, vortex states from this family also saturate the finite-time estimates. This intriguing finding leads to some open questions about the 3D case, namely, whether extreme vortex states with prescribed energy and enstrophy may exhibit a larger growth of enstrophy than the previously found fields in which only enstrophy was fixed and whose growth of enstrophy was too weak to produce a singularity in finite time.

9:18AM M12.00007 Why is the Karman vortex street so stable to the pairing instability? , CRISTOBAL ARRATIA, SAVIZ MOWLAVI, FRANCOIS GALLAIRE, EPFL-LFMI — An infinite double row of staggered point vortices was proposed by von Karman as a simplified model for the alternating vortex street forming in the wake of blunt bodies. This model, however, was found to be always unstable except against infinitesimal disturbances when the aspect ratio of the vortex street has a precise value, a puzzling result in clear contradiction with experience. Several authors including Saffman, Kida and Jimenez studied extensions to Karman's point vortex model, but it turned out that instability for all but a specific value of the parameters is generic in these inviscid models (Jimenez, JFM 1987). Here, we revisit this classical problem from a spatio-temporal instability perspective, which is required for taking into account the propagation speed of the vortex street. We show that the instability of the point vortex model is convective for a large range of parameters, and comparison of the model with different physically relevant cases will be shown. We also consider the absolute/convective nature of the pairing instability in a single row of inviscid point vortices. In both cases we study the effect of confining walls which can be taken into account as an infinite series of image vortices.

9:31AM M12.00008 Coherent Lagrangian vortices: The black holes of turbulence , GEORGE HALLER, ETH Zurich — We discuss a simple variational principle for coherent material vortices in two-dimensional turbulence. Vortex boundaries are sought as closed stationary curves of the averaged Lagrangian strain. We find that solutions to this problem are mathematically equivalent to photon spheres around black holes in cosmology. The fluidic photon spheres satisfy explicit differential equations whose outermost limit cycles are optimal Lagrangian vortex boundaries. As an application, we uncover super-coherent material eddies in the South Atlantic, which yield specific Lagrangian transport estimates for Agulhas rings. We also describe briefly coherent Lagrangian vortex detection to three-dimensional flows.

9:44AM M12.00009 The life of a vortex knot (in experiment)[1] , DUSTIN KLECKNER, MARTIN SCHEELER, University of Chicago, DAVIDE PROMENT, University of East Anglia, WILLIAM T.M. IRVINE, University of Chicago — In recent experiments on linked and knotted vortices in classical fluids, we have found that they undergo a spontaneous change in topology: they untie themselves through a series of local reconnections. This outcome is at odds with the notion that fluid helicity (knottedness) should be conserved, as it should be for a dissipation-less fluid. Remarkably similar behavior is found for simulations of superfluid knots using the Gross-Pitaevskii equation. We will discuss our search for the missing helicity and the possibility of a universal driving mechanism for reconnections in topological vortices.

[1]This work was supported by the National Science Foundation Materials Research and Engineering Centers (MRSEC) Program at the University of Chicago (DMR-0820054) and the Packard Foundation through a Packard fellowship.

Tuesday, November 26, 2013 8:00AM - 9:57AM –

Session M13 Granular Flows IV: Mixing, Segregation and Separation 301 - Kimberly Hill, University of Minnesota

8:00AM M13.00001 Diffusion in linearly sheared granular packing , JOSHUA DIJKSMAN, JIE REN[1], ROBERT BEHRINGER, Duke University — We study shear-induced diffusion in a linearly sheared, dense disordered packing of frictional photoelastic disks. We can track both displacements and rotational motion, and measure interparticle forces obtained from the photoelastic response of the disks. In these experiments, volume fraction and shear amplitude are the control parameters. We probe the non-affine displacements, both in the transient of a single shear deformation, and during cyclic shear. We observe fine structure in the nonffine displacement fields and find that the diffusion anisotropy shows nontrivial dynamics. Additionally, we find that both rotational and translational diffusion increases with density for all but the highest densities.

[1]Currently at Merck & Co.

8:13AM M13.00002 Rate-Based Particle Separation: A Granular "Chromatograph" , DIANA LIEVANO, JOSEPH MCCARTHY, University of Pittsburgh — The effective separation of particles is key to numerous processes and industries handling solid materials. By making an analogy to fluids separations, here we describe a "granular chromatograph" where particle-wall cohesion leads to "adsorption" rates that dictate a particle's traversal down a channel. This adsorption bias leads to differential flow rates of particles that vary in properties, such as size, density, and wetting characteristics. A rate-based separation technique, based on this observation, will be explored.

8:26AM M13.00003 Modeling segregation of bidisperse granular materials: A parametric study[1] , CONOR SCHLICK, YI FAN, PAUL UMBANHOWAR, JULIO OTTINO, RICHARD LUEPTOW, Northwestern University — Predicting segregation and mixing of size bidisperse granular material is a challenging problem with many industrial applications. Using an accurate segregation model based on kinematic properties of the flow that we recently developed, we present a parametric study of segregation of bidisperse granular material in quasi-two-dimensional bounded heaps. The model depends on the Péclet number, Pe, which is the ratio of the advection rate to the diffusion rate, and Λ, which is the ratio of the segregation rate to the advection rate. Both dimensionless parameters depend on the feed rate, the particle size ratio, and the system size. Systematic variation of Λ and Pe demonstrates how the spatial particle configuration depends on the interplay of advection, segregation, and diffusion. At large values of Pe and Λ, segregation dominates and the heap consists of distinct regions of small (upstream) and large (downstream) particles, whereas at low values of Pe and Λ, diffusion dominates which results in a well-mixed heap. Advection plays an important role for large Pe and small Λ and preserves the initial configuration of particles in the feed zone.

[1]Y.F. was funded by The Dow Chemical Company. C.S. was supported by NSF Grant CMMI-1000469.

8:39AM M13.00004 Segregation of Particles by Size and Density in Dense Sheared Flows: Gravity, Temperature Gradients, and Stress Partitioning , DANIELLE TAN, KIMBERLY HILL, St. Anthony Falls Laboratory, Department of Civil Engineering, University of Minnesota — In sheared mixtures of different-sized (same density) particles modestly larger particles tend to go up (toward the free surface), and the smaller particles, down, commonly referred to as the "Brazil-nut problem" or "kinetic sieving." If the larger particles are sufficiently denser than the smaller particles, the segregation reverses. Using theory and simulations, we have recently shown that the segregation fluxes among particles differing *in size only* are driven by two effects: (1) the difference between the partitioning of kinetic and contact stresses among the species in the mixture and (2) a kinetic stress gradient. Specifically, the higher granular temperature of the smaller particles segregates them downward along a kinetic stress gradient toward lower temperatures, and larger particles upward. We adapt the theory to mixtures differing in both size and density and use simulations to show that when the larger particles are sufficiently dense, the theory captures the observed segregation reversal through a reversal in the relative granular temperature born by the two species. In other words, with increasing material density, the larger particles bear increasing fractions of the local kinetic stresses, and the segregation reverses as the larger particles bear a higher fraction than their local concentration in the mixture.

8:52AM M13.00005 Modeling segregation of bidisperse granular materials: Model development[1] , YI FAN, CONOR SCHLICK, PAUL UMBANHOWAR, JULIO OTTINO, RICHARD LUEPTOW, Northwestern University — Predicting segregation of size bidisperse granular materials is a challenging problem. In this talk, we present a theoretical model that captures the interplay between advection, segregation, and diffusion. The fluxes associated with these three driving factors depend on the underlying kinematics, whose characteristics play key roles in determining final particle segregation configurations. Unlike previous models of segregation, our model uses parameters based on kinematic measures instead of arbitrarily adjustable fitting parameters. This permits the theoretical prediction of species concentration within the entire flowing layer as particles segregate in the depth direction while they flow downhill. The model achieves quantitative agreement with both experimental and DEM simulation results when applied to quasi-two-dimensional bounded heaps, and can be readily adapted to other flow geometries.

[1]Y.F. was funded by The Dow Chemical Company. C.P.S. was supported by NSF Grant CMMI-1000469.

9:05AM M13.00006 Flow modulation based control of granular stratification in heaps , PAUL B. UMBANHOWAR, YI FAN[1], DAVID MCDONALD[2], JULIO M. OTTINO, RICHARD M. LUEPTOW, Northwestern University — Gravity driven flows of initially mixed granular media composed of non-monodisperse particles spontaneously segregate for a wide range of particle and flow parameters. For heaps of size-bidisperse particles formed in the quasi-two-dimensional geometry of a vertical Hele-Shaw cell, segregation is in the form of stratified layers of large and small particle-rich bands that are nominally parallel to the free surface of the heap. Stratification occurs at low fill rates where flow down the heap manifests as a series of intermittent and irregularly sized avalanches. This non-steady flow causes variation in stratum thickness and streamwise extent. In this talk we describe how temporal modulation of the fill rate can generate ordered strata at high fill rates. In particular, we show how, for a duty cycle variation of the flow rate, the modulation parameters determine the wavelength and streamwise extent of the layers. We explain our results in terms of the dependence of the dynamic repose angle on flow rate. Finally we describe how the upstream extent of the strata increases with decreasing gap width and is related to the jamming probability of the large particles.

[1]Y.F. was funded by The Dow Chemical Company.
[2]D.M. acknowledges summer support from NSF Grant CMMI-1000469.

9:18AM M13.00007 Passive Separation of Granular Materials , JOSEPH MCCARTHY, DIANA LIEVANO, University of Pittsburgh — Despite its industrial importance, particle separation techniques remain typically quite "low tech" and often are energy-intensive (e.g., sieving) or environmentally unfriendly (e.g., froth floatation) or both. Rate-based separation processes, on the other hand, represent a unique approach to particle separation that has the potential to be more flexible, more efficient, and more environmentally friendly than existing "low tech" techniques. In the present paper, we highlight a passive granular separation technique, where particles of differing properties flow through a device often called a Galton board. In this type of device, the gravity-driven flow of particles down an inclined plane causes collisions between the particles and distributed pegs along the board. Collisions between particles as well as between particles and pegs results in a diffusion-like motion of particles perpendicular to the flow. The extent of separation (i.e., how far one type of particle is removed from another) depends on the different distances traversed by the two types of particles and, ultimately, on the collision rate and energy dissipation for particle-peg events. A simple theory, based on statistics and single-collision mechanics, will be set forth for comparison with our results.

9:31AM M13.00008 Exotic patterns and convection control in a vibrated bed of binary granular mixtures , MEHEBOOB ALAM, ISTAFAUL ANSARI, Jawaharlal Nehru Centre for Advanced Scientific Research — Experiments have been carried out in a harmonically shaken quasi-2D bed of glass and steel particles for a wide range of shaking strengths and relative number fractions of two species. The goal is to understand the role of bidispersity and other control parameters on the resulting pattern formation dynamics and segregation. We report novel patterns displaying the coexistence of sub-harmonic/harmonic and disordered states, and a partial analog of granular Rayleigh-Benard convection. The former patterns bear striking similarities with Chimera-states in the sense that they represent a coexisting state of synchronous and asynchronous patterns. The horizontal segregation of glass and steel particles is responsible for the genesis of such phase-coexisting patterns. We demonstrate a simple recipe to control "buoyancy-driven" granular convection.

9:44AM M13.00009 Suppression and emergence of granular segregation under cyclic shear[1] , MATT HARRINGTON, University of Maryland, JOOST H. WEIJS, University of Twente, WOLFGANG LOSERT, University of Maryland — Heterogeneous mixtures of granular materials have a tendency to segregate under various dynamics disturbances, including shear. While several models have been proposed for segregation in various contexts, there is still much to learn about the mechanisms of shear-induced segregation, particularly at the particle-scale. We have performed experiments on a three dimensional (3D) bidisperse mixture in a split-bottom geometry, under both steady and oscillatory shear. The Refractive Index Matched Scanning technique captures dynamics within the full 3D system. While the pile continuously segregates under steady shear, we find that the cyclically driven system either remains mixed or segregates slowly, depending on shear amplitude. We also characterize the segregating and non-segregating regimes by determining local reversibility with respect to space and structure, as well as observing the emergence of a convective flow field.

[1]Support from NSF and Defense Threat Reduction Agency

Tuesday, November 26, 2013 8:00AM - 9:44AM –

Session M14 Rotating Flows I 302 - Yogesh Jaluria, Rutgers University

8:00AM M14.00001 Small Ekman number heat transport in low Prandtl number rotating thermal convection , ROBERT ECKE, Los Alamos National Laboratory, JOSEPH NIEMELA, International Center for Theoretical Physics — Heat transport in rotating convection is a complex combination of buoyancy, rotation, and fluid nonlinearity. We report experimental measurements of heat transport in rotating convection with cryogenic helium gas having a Prandtl number $Pr = 0.7$. The convection cell is cylindrical with aspect ratio $\Gamma = 1/2$, and the range of explored control parameters, Rayleigh number Ra and Ekman number Ek, is $4 \times 10^9 < Ra < 4 \times 10^{11}$ and $2 \times 10^{-7} < Ek < 3 \times 10^{-5}$ (corresponding to $0.07 < Ro < 5$). We determine the crossover from buoyancy-dominated convection where rotation plays no measurable role in the heat transport to rotation-influenced convection in which the decrease in the heat transport contribution is no greater than 20% of the non-rotating value. We also determine the crossover conditions $Ra_t = 0.5RaEk^{-7/4}$ from the rotation-influenced state to a regime of geostrophic turbulence where normalized heat transport Nu varies roughly linearly in Ra as opposed to the $Ra^{1/3}$ scaling of the rotation-free state. An overall phase diagram of rotating convection in the space of Ra/Ra_c and Ek is proposed for a range of Pr from 0.7 to 6 by combining our results with other data available in the literature.

8:13AM M14.00002 Rotating thermal convection at low Prandtl numbers[1] , STEPHAN WEISS, GUENTER AHLERS, Department of Physics, University of California, Santa Barbara, USA — We present experimental results for rotating thermal convection in a cylindrical cell of aspect ratio $\Gamma \approx 1$ and Prandtl number Pr ≈ 0.7. This value of Pr is relevant to atmospheric convection. By using different compressed gases, we covered the Rayleigh-number range from 6×10^7 to 2×10^{10}. We investigated the transported heat, expressed in terms of the Nusselt number, as well as the sidewall temperature, as a function of the dimensionless rotation rate which we expressed in terms of the inverse Rossby number $1/Ro = 2\Omega/\sqrt{\alpha g \Delta T/L}$. For small Ra we found an increase of Nu with rotation that reached values about 1.5% larger than Nu without rotation. This heat-transport enhancement is significantly smaller than it is for larger Pr, since Ekman pumping cannot efficiently transport warm (cold) fluid from the bottom (top) boundary layer. Numerical simulations by Stevens et al. (NJP Vol. 12, 075005 (2010)) did not resolve any Nusselt-number enhancement for our Pr and Ra numbers. Optical access via shadowgraphy allowed us to study how cold plumes became twisted and formed vortices - a precondition for Ekman pumping.

[1]SW acknowledges support by the Deutsche Forschungsgemeinschaft. This work was supported by the U.S. National Science Foundation through Grant No. DMR11-58514.

8:26AM M14.00003 Ekman and Taylor Vortices' Destruction and Mixing Enhancement in a Taylor–Couette System With Free Surface , HAMID OUALLI, HICHEM BELKADI, ALI ABDELALI, École Militaire Polytechnique, Algiers, Algeria, AHCENE BOUABDALLAH, Université des Sciences et de la Technologie Houari Boumediene, Algiers, Algeria, MOHAMED GAD-EL-HAK, Virginia Commonwealth University, Richmond, Virginia, USA — Suppression of Ekman and Taylor vortices is desirable in several industrial processes such as cylindrical crystal growth and osmotic/photonic water purification. Inhomogeneities are undesired for the former, and enhanced mixing is sought for the latter. An active flow control strategy is employed to obliterate vortices in a Taylor–Couette flow, and is studied experimentally and numerically. The inner cylinder rotates while the outer one is stationary. The gap between the cylinders is not completely filled with liquid, and thus a free surface is formed below the upper stationary end-cap. The control consists of effecting minute radial pulsatile motion of the inner cylinder cross-section. The superimposed modulations combined with the free surface dynamics suppress both the Ekman and Taylor vortices. Complete destruction of either type of vortices occurs at different pulsatile frequencies, requiring one order of magnitude higher frequency to obliterate the Ekman vortex. When eliminated, fluid particles are no longer trapped within the Ekman or Taylor vortices. This yields significant increase in the axial and azimuthal velocity fluctuations, which results in enhanced flow mixing.

8:39AM M14.00004 Stability of the Taylor-Couette flow under a radial thermoelectric body force[1] , HARUNORI YOSHIKAWA, INNOCENT MUTABAZI, OLIVIER CRUMEYROLLE, ANTOINE MEYER, Laboratoire Ondes et Milieux Complexes (UMR 6294 CNRS - Universite du Havre) — A circular Couette flow developed between coaxial two infinite-length cylinders is considered in the case where only the inner cylinder is rotating. A radial temperature gradient and a radial electric field are applied to the flow, their coupling resulting in the dielectrophoretic body force density. This thermoelectric force can stabilize and destabilize the flow, depending on the heating direction. The critical Taylor number, wavenumber and frequency are determined for a wide range of control parameters. The mechanism behind the instability will be discussed.

[1]This work benefited from a financial support from CNES (French Space Agency), the CPER-Haute Normandie under the program THETE and from the FEDER.

8:52AM M14.00005 Secondary Floquet modes of instability in Taylor-Couette flow with axial and radial through-flows , DENIS MARTINAND, ERIC SERRE, LM2P2, Aix Marseille Université - CNRS - Centrale Marseille, RICHARD LUEPTOW, Department of Mechanical Engineering, Northwestern University — Injecting a fluid between a fixed outer impermeable cylinder and concentric rotating permeable inner one and driving it axially is a set-up used in some filtration devices or enzymatic reactors, where the rotation of the inner cylinder promotes mixing or prevents accumulation processes. This set-up can be seen as a Taylor-Couette flow with superimposed axial and radial through-flows and a precise knowledge of the flow structures at stake is a prerequisite for improving these devices. We address the instabilities observed after the laminar flow of a pure, Newtonian solvent has undergone its first two transitions. Previous linear stability analysis has shown that critical convective instabilities take the form of travelling toroidal vortices, turning to helical vortices as the axial flow is increased. Moreover, a weakly non-linear analysis have shown that this primary transition can be subcritical as the radial flow is increased. Based on these previous results, the stability of these primary modes is studied by Floquet analysis. Depending on the strength of the axial and radial flows, harmonic or subharmonic secondary modes are found to be the most dangerous ones. The analytical results are compared to direct numerical simulations using a pseudo-spectral method.

9:05AM M14.00006 Experimental Study of the Flow in a Rotating CVD Reactor , SUN WONG, JIANDONG MENG, YOGESH JALURIA, Rutgers University — An experimental model is developed to study the rotating, vertical, impinging chemical vapor deposition reactor. Deposition occurs only when the system has enough thermal energy. Therefore, understanding the fluid flow and thermal characteristics of the system would provide a good basis to model the thin film deposition process. The growth rate and the uniformity of the film are the two most important factors in the CVD process and these depend strongly on the flow and the thermal transport within the system. Operating parameters, such as inflow velocity, susceptor temperature and rotational speed, are used to create different design simulations. Fluid velocities and temperature distributions are recorded to obtain the effects of different operating parameters. Velocities are recorded by using a rotameter and a hot wire anemometer. The temperatures are recorded by using thermocouples and an infrared thermometer. The effects of buoyancy and rotation are examined. The expermental study is also coupled with a numerical study for validation of the numerical model and to expand the domain. Comparisons between the two models are presented, indicating fair agreement. The numerical model also includes simulation of Gallium Nitride (GaN) thin film deposition. This simulation thus includes mass transport and gas kinetics, along with the flow and heat transfer within the system. A three dimensional simulation is needed due to the rotation of the susceptor. The results obtained as well as the underlying fluid flow phenomena are discussed.

9:18AM M14.00007 How barotropic and stable are differential-rotation cylindrical flows?[1] , GREGORY SHEARD, TONY VO, Department of Mechanical and Aerospace Engineering, Monash University, LUCA MONTABONE, Atmospheric, Oceanic and Planetary Physics, University of Oxford — In rotating cylindrical containers it is possible to generate a highly depth-independent vertical shear layer, akin to the layers studied by Stewartson theoretically in 1957, by driving the inner radial part of the fluid at a different speed through rotation of disks embedded in the top and base of a fluid-filled enclosure. This configuration finds laboratory application in the study of shear layers in rotating flows motivated by geophysical flows such as planetary polar vortices and terrestrial hurricanes. We combine high-order axisymmetric computations with a linear stability analysis and three-dimensional simulation to characterize regimes of depth-independent ("barotropic") flow, and the modes by which both axial (depth-dependent) and azimuthal symmetry are broken in the system. Azimuthal instability produces striking symmetrical polygonal patterns closely resembling patterns seen in atmospheric polar vortices.

[1]Supported by the Australian Research Council through Discovery Grant DP120100153

9:31AM M14.00008 Near-field flow characterization of isothermal coaxial swirling jet , SANTHOSH R, ANKUR MIGLANI, Indian Institute of Science, BHASWATI CHOUDHURY, National Institute of Technology, Karnataka, SAPTARSHI BASU, Indian Institute of Science — The present experimental investigation concerns study of hydrodynamic instability resulting from vortex breakdown in a coaxial type atmospheric swirl burner. Transition from the first occurrence of pre-vortex breakdown (Pre-VB) flow reversal to a fully-developed central toroidal recirculation zone (CTRZ) is studied for a range of swirl number S =0.592 to 0.801.The swirl number was varied progressively by decreasing the mass flow of center jet stream. The physics of the transition is detailed based on modified Rossby number (Ro_m) effect. The decrease in Ro_m across the transition (from an initial 3.15 to 0.02) facilitated the penetration of swirl towards center jet, widening the zone of swirl influence. 2D PIV technique employed in meridional and horizontal planes provided rich insight into the existent dynamics. Transverse plane flow field examination at various axial stations revealed the rigid body rotation characteristic of recirculation flow patterns. Although Rayleigh's criteria for centrifugally unstable flow was satisfied by these coherent structures, the transformation was accompanied by a transition in streamwise vorticity from that dominated by centrifugal forces to the solid body core supporting inertial waves.

Tuesday, November 26, 2013 8:00AM - 10:10AM –

Session M16 Microfluids: General 304 - A. Terrence Conlisk, Ohio State University

8:00AM M16.00001 Analysis of bolus formation from the micropipette ejection systems , DIWEN MENG, PARISA MIRBOD, Clarkson University — Ejection of drugs from micropipettes has significant applications in biomedical research and clinical studies, however little is known about the dynamics of the process involved. The experimental results show that micropipette ejection systems operate in a tip Reynolds number (Ret). A series of experiments was performed from a micropipette to visualize the shape of the droplet. The observations led to the following conclusions: a) A nearly spherical bolus, closely corresponding to Sampson flow through a circular orifice, could be achieved provided at Ret$<$0.05 b) Pear-like bolus distortions are observed at a Ret as small as 0.1. (d) Large distortions are observed at Ret = 0.5 and (e) for Ret $>$ 1 an axial jet develops. Consequently, the transition point between the flow domains represents an important operating point. In this research, laminar is demarcated from turbulent regime by studying the influence of the various material and process parameters on the transition point. Three-dimensional numerical simulations on bolus formation and growth with different tip diameter were investigated and the results were validated with the experimental observations. Effects of fluid physical properties, operation conditions and tip exit size on bolus behavior were also analyzed.

8:13AM M16.00002 Diffusion-limited current to an ion-selective membrane: The role of water splitting and an extended space charge region , CHRISTOFFER P. NIELSEN, HENRIK BRUUS, Department of Physics, Technical University of Denmark, DENMARK — The study of ion-transport across an ion-selective membrane or to an ion-selective surface has found numerous applications in e.g. dialysis, desalination and electrochemistry. The classical 1D LEN (Local Electroneutrality) modeling of the problem has however proven to fall short in many ways, since neither the effect of a finite space charge or the influence of water ions (hydronium and hydroxide) is accounted for in this model. In this work we use a simple model assuming local equilibrium of the water dissociation reaction to model salt and water-ion transport across an ion-selective membrane. The developed numerical and analytical models include the effect of an extended space charge region, and yield current voltage curves and water-ion current versus salt ion current curves which are in qualitative agreement with experimental results. As a result of the analysis a number of simple scaling laws are derived. These are useful for characterizing systems with concentration polarization and allow for easy experimental testing of the model.

8:26AM M16.00003 Deionization shocks in flat and thin microchannels , SHIMA ALIZADEH, MATHIAS B. ANDERSEN, ALI MANI, Department of Mechanical Engineering, Stanford University — We have investigated dynamics of deionization shocks in flat and thin microchannel using two different approaches: (1) extension of Mani and Bazant's simple model [PRE 2011] to two-dimensions, and (2) development of a height-averaged model from tabulated solutions of the Poisson-Boltzmann equation. The latter model is more accurate since it captures both thin and overlapped double-layer regimes as well as diffusion-osmotic flows. Both models describe ion transport and deionization shock dynamics in two dimensional space corresponding to the transverse flat dimensions. We compare prediction of these models for shock profile, speed and dynamical response, as well as onset conditions for hydrodynamic instability of deionization shocks. The outcome of this study has applications in deionization processes in lab-on-a-chip systems as well as porous microstructures.

8:39AM M16.00004 On Taylor dispersion in liquid-cooled electronics applications[1] , B.S. TILLEY, Worcester Polytechnic Institute — We are interested in extending classical asymptotic approaches to allow for the spatial pattern wavenumber to vary on the macroscale variables and to find how changes in microstructure geometry affect macroscopic properties and transport. To this end, we consider here the thermal transport of a coolant through nonuniformly spaced laminates, as a simple model for heat sinks in electronics. Power is continuously being generated by the laminates, and the local rates of heat transport depend on convection, fluid inertia, buoyancy and Taylor dispersion in the coolant and conduction within both the fluid and the laminates. We find a coupled system of partial differential equations that describe the local microscale temperature and deviations from the Darcy pressure. Microscale values of all of these quantities are known in terms of the solutions to these effective eqautions. We are especially interested in geometries in the laminate spacing which allow for better thermal transport by the coolant for a prescribed power distribution. The choice of the channel geometries depend on the ability to transfer heat from the device to the enviornment, the orientation of the device with respect to gravity, and the available power needed to drive the fluid motion.

[1]This work is supported by a grant from the Air Force Office of Scientific Research, (Award No. FA9550-11-1-0197).

8:52AM M16.00005 Characterization of Heat Transfer in Superhydrophobic Microchannels under Different Wetting Modes , TAE JIN KIM, CARLOS HIDROVO, The University of Texas at Austin — Slip flow in microchannels is known to reduce the wall friction and consequently decreases the pumping power to drive the flow. One method to achieve slip flow is by trapping gas bubbles in the microchannel wall that is highly corrugated. While the use of rough walls to induce friction reduction is attractive, many microfluidic applications involve coupling of heat source in the microchip: the gas pockets may affect the heat transfer from the heaters to the microchannel walls. The purpose of this research is to explore the heat transfer efficiency of microchannels with corrugated surfaces heated from the side walls. The microchannel walls are modified to have an array of micro-trenches arranged transverse to the fluid flow along the axial direction, and a constant water pressure source is used to drive the flow and control the air pocket size. Advective heat transfer is then analyzed between the microchannel inlet and outlet using laser induced thermometry technique. Under identical flow rate conditions, it is expected that 1) the advective efficiency is affected by the degree of wetting of the corrugated walls and that 2) the advective heat transfer is lower for superhydrophobic microchannels with gas pockets trapped in the corrugated walls than those filled with water.

9:05AM M16.00006 Mesoscopic modeling of non-isothermal fluid systems[1] , ZHEN LI, YUHANG TANG, Division of Applied Mathematics, Brown University, BRUCE CASWELL, School of Engineering, Brown University, GEORGE EM KARNIADAKIS, Division of Applied Mathematics, Brown University — The dynamical properties of fluid, including diffusivity and viscosity, are temperature-dependent and can significantly influence the flow dynamics in non-isothermal systems. To capture the correct temperature-dependence of a fluid, an energy conserving dissipative particle dynamics (eDPD) model is developed by expressing the weighting functions of the dissipative force and the random force as functions of temperature. The diffusivity and viscosity of liquid water at various temperatures ranging from 273K to 373K are used as examples for verifying the proposed model. For non-isothermal fluid systems, the present model can predict the diffusivity and viscosity consistent with available experimental data of water at various temperatures. Moreover, an analytical formula for determining the mesoscopic heat friction is proposed. The validation of the formula is confirmed by reproducing the experimental data in Prandtl number of liquid water at various temperatures. The proposed method is demonstrated in water but it can be readily extended to other liquids.

[1]Supported by the new DOE Center on Mathematics for Mesoscopic Modeling of Materials (CM4) and an INCITE grant.

9:18AM M16.00007 Modelling the extrusion of preforms for microstructured optical fibres , HAYDEN TRONNOLONE, YVONNE STOKES, University of Adelaide, DARREN CROWDY, Imperial College London — Owing to a novel design, microstructured optical fibres (MOFs) promise the realisation of fibres with effectively any desired optical properties. MOFs are typically constructed from glass and employ a series of air channels aligned along the fibre axis to form a waveguide. The construction of MOFs by first extruding a preform and then drawing this into the final fibre has the potential to produce fibres on an industrial scale; however, this is hindered by a limited understanding of the fluid flow that arises during this process. We focus on the extrusion stage of fabrication and discuss a model of the fibre evolution based upon complex-variable techniques. The relative influence of the various physical processes involved is discussed, along with limitations of the model.

9:31AM M16.00008 Stresses due to Relative Sliding between Particles Surrounded by an Electrolyte Solution with Application to Lithium-Ion Batteries[1] , CONG ZHANG, A.T. CONLISK, The Ohio State University — Mechanical stresses in the solid phase of the electrodes within lithium-ion batteries have been the subject of much work recently with the emphasis on the stresses induced by lithium insertion to or extraction from the active solid material. The particles within lithium-ion battery electrodes can undergo relative motion with relative velocities of different magnitudes and directions. One mode of the relative motion, resembling the slider bearing motion, manifests itself as two particles sliding relative to each other within an electrolyte solution. The electrolyte solution within the narrow pores between the particles is the medium through which the particles interact with each other. The effect of the electrolyte solution is not conventionally considered. The relative motion of the particles induces significant pressures. The primary objective of this work is to develop a model based on the lubrication approximation to investigate the magnitude and direction of the stresses induced by this sliding motion. Other applications in the biomedical field are also discussed.

[1]Supported by DOE Graduate Automotive Technology Education (GATE) and OSU Center for Automotive Research.

9:44AM M16.00009 Stresses due to Squeeze Flow between Particles Surrounded by an Electrolyte Solution with Application to Lithium-Ion Batteries[1] , A.T. CONLISK, CONG ZHANG, The Ohio State University — Large stresses are induced during lithium-ion battery charging and discharging, termed intercalation and deintercalation stresses. Current models of the stresses in lithium-ion batteries in the literature seldom consider the influence of the interaction between the particles within the electrodes on the stress distribution. The particles within lithium-ion battery electrodes can undergo relative motion with relative velocities of different magnitudes and directions. One important mode of motion manifests itself as two particles approaching each other. The interaction is mediated by the electrolyte between the particles. The relative motion of the particles induces significant pressures and the primary objective of this work is to propose a source of mechanical stresses as a consequence of the dynamic squeezing motion as opposed to a static environment considered in the battery literature. Other applications in the biomedical field are also discussed.

[1]Supported by DOE Graduate Automotive Technology Education (GATE), OSU Center for Automotive Research and OSU NSEC Center for the Affordable Nanoengineering of Polymeric Biomedical Devices.

9:57AM M16.00010 PIV measurements of the transient fluid flow due to the adsorption of particles[1] , NAGA MUSUNURI, PRARTHITH B. SHAH, IAN S. FISCHER, PUSHPENDRA SINGH, New Jersey Institute of Technology — The particle image velocimetry (PIV) technique is used to study the physics of particle adsorption and the spontaneous dispersion of powders that occurs when particles come in contact with a fluid-liquid interface. The dispersion can occur so quickly that it appears explosive, especially for small particles on the surface of mobile liquids like water. The measurements show that the adsorption of a spherical particle causes an axisymmetric streaming flow about the vertical line passing through the center of the particle. The fluid directly below the particle rises upward, and near the surface, it moves away from the particle. The flow, which develops within a fraction of second after the adsorption of the particle, persists for several seconds. The flow strength, and the volume over which it extends, decrease with decreasing particle size. The streaming flow induced by the adsorption of two or more particles is a combination of the flows which they induce individually.

[1]We would like to thank NSF.

Tuesday, November 26, 2013 8:00AM - 10:10AM –

Session M17 Biofluids: Locomotion VIII - Bats and Butterfly Flight 305 - Haibo Dong, University of Virginia

8:00AM M17.00001 Understanding the energy economy of a batoid-inspired flexible fin[1] , FLORIAN H.J. BREMER, STEFANO CHIAZZA, Princeton University, ALEXANDER J. SMITS, Princeton University and Monash University — Batoid-inspired autonomous underwater vehicles are interesting in that they offer the promise of fast and efficient motion. To investigate the effects of flexibility of the pectoral fins on the energy economy, free-swimming experiments are conducted an artificial fin in flapping motion. The experiments are conducted by initiating a flapping motion in the stationary fin, and by allowing the fin to accelerate to its free-swimming speed while keeping the amplitude and frequency of the actuation constant. The energy economy is derived by continuously measuring velocity and power input. Comparisons are then made among fins of varying flexibility to find the optimal energy economy.

[1]This work was supported by the ONR through MURI Grant N00014-08-1-0642 (Program Manager: Dr. Bob Brizzolara).

8:13AM M17.00002 The effect of aspect ratio on the generation of lift and drag of a compliant membrane flapping wing , COSIMA SCHUNK, KRISTEN MICHAELSON, TRISTAN PAINE, SHARON M. SWARTZ, KENNETH S. BREUER, Brown University — Aspect ratio is frequently used to describe differences between the large variety of bat wing shapes. Bats with high aspect ratio wings are expected to fly with a high efficiency and to have superior lift-to-drag ratios. In contrast, bats with lower aspect ratio wings are thought to exhibit higher maneuverability. However, those assumptions are often based on theoretical models based on fixed wing aerodynamic theory. To examine the performance of highly compliant wings with different aspect ratios in flapping flight, we measure lift and drag generated by a mechanical flapping wing. A two degree of freedom shoulder joint allows for independent control of flapping amplitude and wing sweep. Several bat-like wings with different aspect ratios but identical surface area were built, and tested in a wind tunnel, and the variations of lift and drag over the wingbeat cycle are measured over a flapping frequency range of 2 - 10 Hz.

8:26AM M17.00003 Deconstructing the Essential Elements of Bat Flight[1] , DANESH TAFTI, Virginia Tech, KAMAL VISWANATH, Naval Research Laboratory, NAGENDRA KRISHNAMURTHY, Virginia Tech — There are over 1000 bat species worldwide with a wide range of wing morphologies. Bat wing motion is characterized by an active adaptive three-dimensional highly deformable wing surface which is distinctive in its complex kinematics facilitated by the skeletal and skin membrane manipulation, large deviations from the stroke plane, and large wing cambers. In this study we use measured wing kinematics of a fruit bat in a straight line climbing path to study the fluid dynamics and the forces generated by the wing using an Immersed Boundary Method. This is followed by a proper orthogonal decomposition to investigate the dimensional complexity as well as the key kinematic modes used by the bat during a representative flapping cycle. It is shown that the complex wing motion of the fruit bat can mostly be broken down into canonical descriptors of wing motion such as translation, rotation, out of stroke deviation, and cambering, which the bat uses with great efficacy to generate lift and thrust.

[1]Research supported through a grant from the Army Research Office (ARO). Bat wing kinemtaics was provided by Dr. Kenny Breuer, Brown University.

8:39AM M17.00004 Lift and thrust generation by a butterfly-like 3D flapping wing model , KOSUKE SUZUKI, TAKAJI INAMURO, Dept. Aeronautics and Astronautics, Kyoto University — The flapping flight of tiny insects such as a butterfly is of fundamental interest not only in biology itself but also in its practical use for the development of micro air vehicles. It is known that a butterfly flaps downward for generating lift force and backward for generating thrust force. In this study, we consider a simple butterfly-like 3D flapping wing model whose body is a thin rod, wings are rigid and rectangular, and wing motion is simplified. We investigate the lift and thrust generation by the butterfly-like flapping wing model by using the immersed boundary-lattice Boltzmann method. Firstly, we compute the lift and thrust forces when the body of the model is fixed for Reynolds numbers in the range of 50 - 1000. In addition, we evaluate the supportable mass for each Reynolds number by using the computed lift force. Secondly, we simulate the free flight where the body can move translationally but cannot rotate. As results, we find that the evaluated supportable mass can be supported even in the free flight, and the wing model with the mass and the Reynolds number of a fruit fly can go upward against the gravity. Finally, we simulate the effect of the rotation of the body. As results, we find that the body has a large pitching motion and consequently gets off-balance.

8:52AM M17.00005 A Simple Analytical Model for Batoid Wake topology and Propulsive Forces[1] , PABLO VALDIVIA Y ALVARADO, KARTHIK SRIVATSA, Singapore MIT Alliance for Research and Technology — Batoids swim by forcing waves along their large pectoral fins. These waves determine the topology of the shed wakes and the resulting propulsive forces. An understanding of the relation between fin kinematics and wake topology is essential to control vehicles that use batoid-like fin propulsion. Simulations of the fluid-structure interactions during fin motions provide information of the changes in wake topology and the propulsive forces that result with variations in fin kinematics. However, simulations require computing power usually not available in mobile robots and cannot be used for real time control. An alternative is to develop simple qualitative models whose errors can be compensated by closed loop feedback controllers. Here we describe an analytical model that can be used to predict wake geometry and resulting propulsive forces in batoid-like fins. The model incorporates important fin kinematic parameters such as wave number, amplitude envelope, and flapping frequency. Dye flow visualization and particle image velocimetry along with force measurements confirm the model applicability to batoid-like fin propulsion.

[1]This work was funded in whole or in part by the Singapore National Research Foundation (NRF) through the Singapore-MIT Alliance for Research and Technology (SMART)

9:05AM M17.00006 Investigation into the Role of Dragonfly Wing Flexibility During Passive Wing Pitch Reversal[1] , YOUSAF BAJWA[2], VENTRESS WILLIAMS[3], YAN REN[4], HAIBO DONG[5], University of Virginia, FLOW SIMULATION RESEARCH GROUP TEAM — Wing deformation is a characteristic part of flapping wing flight. In dragonflies, a torsion wave can be observed propagating from the tip to the root during stroke reversal. In this paper, we utilize high-speed photogrammetry and 3d surface reconstruction techniques to quantify wing deformation and kinematics of a dragonfly. We then use finite elements in the absolute nodal coordinate formulation to estimate strain energy in the wing during wing pitch reversal. We use this data to analyze the role of wing structure in facilitating wing rotation and bringing about the characteristic torsion wave. The influence of the elastic force in facilitating wing rotation is then compared with inertial and aerodynamic forces as well. A quantitative look into the variation of strain energy within the insect wing during wing rotation could lead to more efficient design of dynamic wing pitching mechanisms.

[1]Supported by NSF CBET-1343154
[2]Undergraduate Student, ynb5fb@virginia.edu
[3]Undergraduate Student, vcw5fw@virginia.edu
[4]Graduate Student, alanrenyan@gmail.com
[5]Research Advisor, hd6q@virginia.edu

9:18AM M17.00007 Analysis of Dragonfly Take-off Mechanism: Initial Impulse Generated by Aerodynamic Forces[1] , RUIJIE ZHU[2], AYODEJI BODE-OKE[3], YAN REN[4], HAIBO DONG[5], University of Virginia, FLOW SIMULATION RESEARCH TEAM — Take-off is a critical part of insect flight due to not only that every single flight initiates from take-off, but also that the take-off period, despite its short duration, accounts for a relatively large fraction of the total energy consumption. Thus, studying the mechanism of insect take-off will help to improve the design of Micro Air Vehicles (MAVs) in two major properties, the success rate and the energy efficiency of take-off. In this work, we study 20 cases in which dragonflies (species including *Pachydiplax longipennis*, *Epitheca Cynosura*, *Epitheca princeps* etc.) take off from designed platform. By high-speed photogrammetry, 3-d reconstruction and numerical simulation, we explore how dragonflies coordinate different body parts to help take-off. We evaluate how aerodynamic forces generated by wing flapping create the initial impulse, and how these forces help save energy consumption.

[1]Supported by NSF CBET-1343154
[2]Undergraduate Student
[3]Undergraduate Student
[4]Graduate Student
[5]Research Advisor

9:31AM M17.00008 Force production of a hovering hummingbird[1] , HAOXIANG LUO, JIALEI SONG, Vanderbilt University, TYSON HEDRICK, University of North Carolina at Chapel Hill — A three-dimensional numerical study is performed for a hovering Ruby-throated hummingbird (*Archilochus colubris*) based on an immersed-boundary method. To accurately model the unsteady aerodynamics, realistic 3D wing kinematics is reconstructed from high-speed images of the wing motion filmed at 1000 frames per second, resulting in 25 frames per flapping cycle. A high-resolution grid is employed to resolve the vortices shed from the wing. The results are validated by comparing the spanwise vorticity and circulation with the previous PIV data and also by calculating the average lift. The force production shows significant asymmetry with the downstroke producing lift 2.6 times as high as the upstroke, despite a nearly horizontal stroke plane. The total power consumption is around 55 W/kg, which is twice of previous estimate. In this presentation, we will discuss several mechanisms that lead to the force asymmetry, including the drag-based lift and the leading-edge vortex behavior. We will also address the role of wing-wake interaction, which appears to be different for the hummingbird than some of the insects such as fruit flies.

[1]Supported by NSF (No. CBET-0954381)

9:44AM M17.00009 Investigation of sharp-turning mechanism of damselfly via motion kinematics and vortex dynamics , YU-CHEN TSAI, JING-TANG YANG, Department of Mechanical Engineering, National Taiwan University — Damselflies often perform unconventional backward flight to make sharp turns while changing direction in forward flight. The mechanism of rapid transition between forward and backward flight of the free-flying damselfly (Psolodesmus mandarinus) is experimentally investigated in this study. The flapping kinematics of the damselflies during flight reversal is observed and recorded by using two high-speed cameras. The flow field is examined first and the vortex structure is further analyzed by using particle imaging velocimetry (PIV) technique. The relation between the kinematic parameters of a moving damselfly and its excellent turning ability is revealed; the damselfly makes a sharp turn in merely tens of milliseconds by altering body posture, flapping frequency, and angle of attack. The strong interaction between the wings and the surrounding vortices is proved crucial in producing forces needed for turning. This study provides insights into the maneuvering strategy of flying insects.

9:57AM M17.00010 The mechanism of body rotation in the flapping flight of butterflies , YUEH-HAN FEI, JING-TANG YANG, Department of Mechanical Engineering, National Taiwan University — The aerodynamic effects of the body rotation on the flapping flying of butterflies are experimentally and numerically investigated. We first observe and record the flying motion of a butterfly (Kallima inachus) in free flight, focusing especially on the body rotation, via two high speed video cameras and PIV. The body rotation is found in phase with wing flapping while the abdomen is out of phase with wing flapping. Further, we establish the model of flexible wings of a butterfly and exploit the fluid dynamics analysis via the dynamic mesh technique to study the contribution of body rotation to the lift. The results reveal that the body rotation is capable of strengthening the vortex ring structure and correspondingly enhancing the efficiency of lift production. Our simulation model shows the body rotation contributes 15% of total lift. The results of this study may serve as a useful guide for designing insect-like MAVs in the future.

Tuesday, November 26, 2013 8:00AM - 9:31AM –

Session M18 Biofluids: General VII - Biofilms 306/307 - Knut Drescher, Princeton University

8:00AM M18.00001 A framework to understand cell type transitions in bacterial biofilms , AGNESE SEMINARA, CNRS - Laboratoire de physique de la matière condensée, Nice, France, NAVEEN SINHA, JAMES WILKING, STEPHANE KOELHER, School of Engineering and Applied Sciences, Harvard University, Cambridge, MA, MATTHEW CABEEN, Department of molecular and cellular biology, Harvard University, Cambridge, MA USA, DAVID WEITZ, MICHAEL BRENNER, School of Engineering and Applied Sciences, Harvard University, Cambridge, MA — Bacterial biofilms are colonies of cells that live associated to surfaces and differentiate into different cell types, in response to unknown environmental cues. Similar to the development of multicellular organisms, differentiation happens in reproducible spatio-temporal patterns of gene expression. Why do we see the patterns that we see? Fluorescence microscopy shows that there is a cell lineage specific to biofilms: cells are first motile, they then become matrix producers, and finally they sporulate. We combine this knowledge to the complete space-time distribution of fluorescence to study when and where the transitions among these three cell types arise. We first isolate the effect of growth and expansion on the evolution of the expression profiles to detect the cell type transitions. Based on these data we then elaborate a consistent scenario to explain cell type transitions.

8:13AM M18.00002 Measurement of fluid dynamic loading on staphylococci bacteria bio-film structures using μPIV , ERICA SHERMAN, University of Nebraska - Lincoln, DEREK MOORMEIER, KENNETH BAYLES, University of Nebraska Medical Center, JOHN DAVIDSON, SANGJIN RYU, TIMOTHY WEI, University of Nebraska - Lincoln — Staphylococci bacteria are recognized as the most frequent cause of biofilm-associated infections. Although humans are regularly exposed to these bacteria without consequence, a localized infection can enter the bloodstream and lead to serious infections such as endocarditis, pneumonia, or toxic shock syndrome. The mechanics of staphylococci biofilm formation and dispersion through the bloodstream are not well known. It has recently been observed that under certain flow conditions, bacteria organize in tower-like structures which break and are transported downstream by the flow. The fundamental questions of interest are i) whether or not fluid mechanics plays a role in differentiating between film or tower formation and ii) whether or not the faulty towers are a bio-film propagation mechanism. This talk focuses on the application of μPIV to study this problem. Staphylococcus aureus bacteria were cultured in the Bioflux Fluxion square microchannel of a 65 by 65 um cross section, and subjected to a steady shear rate of 0.5 dynes. μPIV measurements were made to map the flow over and around a biofilm tower structure which occluded approximately 66% of the channel width. Data were recorded around the structure at a series of two dimensional planes, which when stacked vertically show a two dimensional flow field as a function of tower height. Measurements and control volume analysis will be presented quantifying forces acting on these structures.

8:26AM M18.00003 Kinetic theory for actively streaming microtubule suspensions , TONG GAO, Courant Institute of Mathematical Sciences, ROBERT BLACKWELL, MATT GLASER, MEREDITH BETTERTON, Department of Physics and Liquid Crystal Materials Research Center and Biofrontiers Institute, MICHAEL SHELLEY, Courant Institute of Mathematical Sciences — Suspensions of polar biopolymers mixed with molecular motor proteins can exhibit surprising out-of-equilibrium phenomena. In a recent experiment by Sanchez et al., microtubules are driven into collective motion by plus-end walking motor complexes. In experiments where the suspension is confined to a fluid-fluid interface, they find the emergence of distinctive large-scale flows characterized by persistent time-dependence and formation/annihilation of disclination singularities in the nematic order. Here we develop a first-principles kinetic theory to investigate the nonlinear dynamics and pattern formation observed in active microtubule suspensions. We model the active stresses generated by motile microtubules by taking into account the extensile stresses due to both the antiparallel and the parallel microtubule pairs. In a concentrated system, the resultant particle-pair stresses can induce hydrodynamic instabilities, and lead to a large-scale flows. When the suspension is confined to a liquid-liquid interface, we recover much of the dynamics observed in the experiments.

8:39AM M18.00004 Mathematical Modeling of Tear Film Break up Modes and Fluorescent Intensity[1] , JAVED SIDDIQUE, Penn State York, RICHARD BRAUN, Department of Mathematics, University of Delaware, CAROLYN BEGLEY, ADAM WINKELER, School of Optometry, Indiana University, Bloomington IN, PETER E. KING-SMITH, College of Optometry, Ohio State University, Columbus, OH — The purpose of this study is to develop mathematical model for variables of interest in tear film break up (TBU) to compare with experimental images of TBU to better predict local values of tear film osmolarity and fluorescence during and following the TBU. Models are developed for local changes tear film thickness, insoluble surfactant concentration as well as osmolarity and fluorescein concentration inside the tear film. Fluorescence concentration was converted to fluorescent intensity using the expression involving film thickness and the full range of fluorescence as described by Nichols et al (IOVS 2012). The fluorescent intensity response is a primary tool for visualizing the tear film thickness, and it is qualitatively different in the dilute vs concentrated regimes. Computed results over a wide range of fluorescein concentrations show that elevated surfactant concentration or evaporation rate led to thinner regions where TBU first occurs. The model predicts locally elevated concentration of osmolarity within areas of TBU and fluorescence intensity patterns very similar to computed thickness and the observed experimental results. The osmolarity may increase from 50% to 1300% of the isosmolar value, depending sensitively on the corneal permeability.

[1]Supported by Simons Foundation Grant 281839 (JIS), NSF Grant 1022706, NEI Grant 1RO1EY021794 (Begley), and NEI Grant RO1EY17951 (King-Smith)

8:52AM M18.00005 Physical solutions to the public goods dilemma in bacterial biofilms , KNUT DRESCHER, CAREY NADELL, HOWARD STONE, NED WINGREEN, BONNIE BASSLER, Princeton University — Bacteria frequently live in densely populated surface-bound communities, termed biofilms. Biofilm-dwelling cells rely on secretion of extracellular substances to construct their communities and to capture nutrients from the environment. Some secreted factors behave as cooperative public goods: they can be exploited by non-producing cells. The means by which public good producing bacteria avert exploitation in biofilm environments are largely unknown. Using experiments with *Vibrio cholerae*, which secretes extracellular enzymes to digest its primary food source, the solid polymer chitin, we show that the public goods dilemma may be solved by two dramatically different, physical mechanisms: cells can produce thick biofilms that confine the goods to producers, or fluid flow can remove soluble products of chitin digestion, denying access to non-producers. Both processes limit the distance over which enzyme-secreting cells provide a benefit to neighbors, resulting in preferential benefit to nearby clonemates. Our results demonstrate how bacterial physiology and environmental conditions can interact with social phenotypes to influence the evolutionary dynamics of cooperation within biofilms.

9:05AM M18.00006 Coupling Osmolarity Dynamics within Human Tear Film on an Eye-Shaped Domain , LONGFEI LI, R.J. BRAUN, T.A. DRISCOLL, University of Delaware, W.D. HENSHAW, J.W. BANKS, Lawrence Livermore National Laboratory, P.E. KING-SMITH, Ohio State University — The concentration of ions in the tear film (osmolarity) is a key variable in understanding dry eye symptoms and disease. We derived a mathematical model that couples osmolarity (treated as a single solute) and fluid dynamics within the tear film on a 2D eye-shaped domain. The model concerns the physical effects of evaporation, surface tension, viscosity, ocular surface wettability, osmolarity, osmosis and tear fluid supply and drainage. We solved the governing system of coupled nonlinear PDEs using the Overture computational framework developed at LLNL, together with a new hybrid time stepping scheme (using variable step BDF and RKC) that was added to the framework. Results of our numerical simulations show good agreement with existing 1D models (for both tear film and osmolarity dynamics) and provide new insight about the osmolarity distribution over the ocular surface during the interblink.

9:18AM M18.00007 Thin film drainage between pre-inflated capsules or vesicles[1] , MARTIN KEH, JOHANN WALTER, GARY LEAL, University of California, Santa Barbara — Capsules and vesicles are often used as vehicles to carry active ingredients or fragrance in drug delivery and consumer products and oftentimes in these applications the particles may be pre-inflated due to the existence of a small osmotic pressure difference between the interior and exterior fluid. We study the dynamics of thin film drainage between capsules and vesicles in flow as it is crucial to fusion and deposition of the particles and, therefore, the stability and effectiveness of the products. Simulations are conducted using a numerical model coupling the boundary integral method for the motion of the fluids and a finite element method for the membrane mechanics. For low capillary numbers, the drainage behavior of vesicles and capsules are approximately the same, and also similar to that of drops as the flow-independent and uniform tension due to pre-inflation dominates. The tension due to deformation caused by flow will become more important as the strength of the external flow (i.e. the capillary number) increases. In this case, the shapes of the thin film region are fundamentally different for capsules and vesicles, and the drainage behavior in both cases differs from a drop.

[1]Funded by P&G.

Tuesday, November 26, 2013 8:00AM - 9:31AM –

Session M19 Boundary Layers VII: Wind Turbine Interaction 310/311 - James Brasseur, The Pennsylvania State University

8:00AM M19.00001 Turbulence-driven power fluctuations on a wind turbine: characterization in the spectral domain , LEONARDO CHAMORRO, NICOLAS TOBIN, HYUN J. KIM, JIN T. KIM, University of Illinois at Urbana-Champaign — Power and loading fluctuations experienced by wind turbines are mostly driven by the turbulent characteristics of the incoming flow, which limit their life span. Understanding the complex relation between wind turbine(s) and flow unsteadiness is key for the development of advanced controls and also in structural design. In this field study, we investigate the response of a 1kW wind turbine under various inflow conditions. The research is performed in the RE-TE Wind Energy Field Station of the University of Illinois. Synchronous measurements of the three velocity components of the incoming flow, turbine power and rotational speed of the rotor are acquired at a temporal resolution that includes the majority of the scales relevant for the problem. An array of sonic anemometers is used to characterize the incoming flow in the vicinity of the wind turbine. Insights on the scale-to-scale interaction between flow and the turbine are obtained as well as the linkage between their spectral structures. A comparison with a wind-tunnel experiment and full-scale setup suggest an apparent universal behavior of the spectral structure of the wind turbine power.

8:13AM M19.00002 Streamwise evolution of statistical events and the triple correlation in a model wind turbine array[1] , KYLE VIESTENZ, RAÚL BAYOÁN CAL, Portland State University — Hot-wire anemometry data, obtained from a wind tunnel experiment containing a 3×3 wind turbine array, are used to conditionally average the Reynolds stresses. Nine profiles at the centerline behind the array are analyzed to characterize the turbulent velocity statistics of the wake flow. Quadrant analysis yields statistical events occurring in the wake of the wind farm, where quadrants 2 and 4 produce ejections and sweeps, respectively. A balance between these quadrants is expressed via the ΔS_o parameter, which attains a maximum value at the bottom tip and changes sign near the top tip of the rotor. These are then associated to the triple correlation term present in the turbulent kinetic energy equation of the fluctuations. The development of these various quantities is assessed in light of wake remediation, energy transport and possess significance in closure models.

[1]National Science Foundation: ECCS-1032647

8:26AM M19.00003 The effect of two-bladed and three-bladed wind turbine rotors on fluxes of kinetic energy , DALTON MCKEON, ANDREW NEWMAN, Texas Tech University, MATTHEW MELIUS, RAUL CAL, Portland State University, LUCIANO CASTILLO, Texas Tech University — As energy is extracted by wind turbines in an array, the main mechanism entraining energy into the flow within the array has been shown to be turbulent kinetic energy flux. Experiments showing this relationship have utilized three-bladed rotors. The goal of this study is to describe fluxes of kinetic energy in arrays utilizing two-bladed and three-bladed rotors. In a wind tunnel, two 3 X 4 arrays of model wind turbines were exposed to neutrally stratified conditions, with one array using two-bladed rotors and the other using three-bladed rotors. Both arrays had three turbines with 3D spacing in the spanwise direction and four turbines with 6D spacing in the streamwise direction. Each rotor had a diameter of 12 cm. The power coefficient was matched so that the non-dimensional rate of energy extraction was the same for both arrays. Data was collected along the centerline of both arrays using PIV with vertical data planes parallel to the streamwise direction. A control volume bounded by the rotor top tip and bottom tip is considered and fluxes of kinetic energy are compared for two-bladed and three-bladed arrays. Preliminary results show similar trends for the fluxes in both arrays, but no direct scaling was found, indicating a more complicated dependence on the number of blades.

8:39AM M19.00004 Flow event classification via conditional statistics of PIV data in a model wind turbine array[1] , DEVIN KNOWLES, RAÚL BAYOÁN CAL, Portland State University — Conditional statistics are employed in the analysis of velocity data taken on a 3x4 model wind turbine array in a scaled wind tunnel experiment. Dual Stereo Particle Image Velocimetry measurements of the incoming and wake flow fields are considered for the wind turbines in the first and fourth row positions along the centerline. Quadrant analysis is applied to the SPIV data for all components of the Reynolds stress tensor $\langle u_i u_j \rangle$. The flow events involving all three components of velocity are identified and characterized using this conditional averaging technique as proposed by Raupach.[2] Particular attention is given to the out of plane component of velocity due to the 3-dimensional nature of the flow in the wind turbine wake.

[1]National Science Foundation: ECCS-1032647
[2]Raupach, M. R., (1981), *J. Fluid Mechanics*, **108**, 363 – 382.

8:52AM M19.00005 The Structure of the Wind Turbine Array/ Atmospheric Boundary Layer Interface , LUCIANO CASTILLO, JENSEN NEWMAN, Texas Tech University — A structure based approach is taken to describe the interface between a scaled wind turbine array in a wind tunnel and the approximate atmospheric boundary layer above it. The principle interest is to identify the various types of structures which exist here and determine 1) the domain of their existence and 2) their role in the process of extracting energy from the mean flow above the wind turbine array. These goals are achieved by computing the proper orthogonal decomposition of particle image velocimetry data collected along the centerline of the scaled wind turbine array and projecting the instantaneous field onto the most energetic modes. The analysis is carried out behind the first four turbines in the array and reveals two distinct regions based on structural differences: The near wake which is dominated by small scale turbulence and the far wake which is dominated by larger scales. It is further shown that the majority of the energy extraction is done by the larger scales in the far wake. Further, it is shown that after the first turbine, the small scale turbulence in the near wake is often less energetic than existing larger scale turbulence in the inflow.

9:05AM M19.00006 Kinetic Energy Transport in a Vertical-Axis Wind Turbine Array[1] , MATTHIAS KINZEL, DANIEL ARAYA, JOHN DABIRI, Caltech — We present experimental results from a full scale array of vertical-axis wind turbines (VAWTs) under natural wind conditions. The wind velocities throughout the turbine array are measured using a portable meteorological tower with seven, vertically-staggered, three-component ultrasonic anemometers. These measurements yield detailed insight into the turbine wakes and the recovery of the flow velocity behind the turbines. Quadrant hole analysis is employed to gain a better understanding of the energy transport at the top and the bottom of the VAWT array. The results are compared to the flow in horizontal-axis wind farms as well as urban and plant canopies. Emphasis is given to the flow physics in the adjustment region of the canopy, i.e. the region where the flow transitions from an atmospheric boundary layer to a canopy flow.

[1]The authors gratefully acknowledge funding from the Gordon and Betty Moore Foundation through Grant 2645, the National Science Foundation Energy for Sustainability program (Grant No. CBET-0725164) and the Office of Naval Research (Grant No. N000141211047)

9:18AM M19.00007 Development of a Scaled Smart Wind Farm , SUHAS POL, AMELIA TAYLOR, DALTON MCKEON, LUCIANO CASTILLO, ISAIAS PEREZ, REN BEIBEI, JIAN SHENG, CARSTEN WESTERGAARD, AKSAK BURAK, GUILLERMO ARAYA, FAZLE HUSSAIN, Texas Texas University — A model wind farm consisting of 3X5 horizontal axis turbines with a rotor diameter of 4 m (to be expanded to 5X20 turbines of 2m diameter) is being developed on TTU campus. Real field turbine wake evolution and interactions will be studied by employing particle image velocimetry. A 10m tower upstream of the wind farm as well as a 200m tower located 500m from the site will be used to characterize the atmospheric condition and its influence on the wake evolution. Of particular interest is the role of coherent structures in the atmosphere and the wake on the downward transport of overhead momentum - hence the effectiveness of the wind farm. From the recorded data episodes of stable, unstable and neutral atmosphere will be conditionally sampled to understand the effect of atmospheric stability on wind farm dynamics. The effect of various turbine-turbine separation and orientation on the downward momentum transport will be studied – quite feasible since the turbine models are portable. In addition to aerodynamic studies the facility we will also test control algorithms.

Tuesday, November 26, 2013 8:00AM - 10:10AM –

Session M20 Instability: General IV 315 - John Cressman, George Mason University

8:00AM M20.00001 Localized convection in a rotating system , CEDRIC BEAUME, HSIEN-CHING KAO, EDGAR KNOBLOCH, UC Berkeley, ALAIN BERGEON, Universite de Toulouse — We study two-dimensional spatially localized convection in a horizontal fluid layer rotating around the vertical and heated from below [1]. With stress-free boundary conditions stationary spatially localized convection is present [2]. These states are embedded in a background shear layer and lie on a pair of intertwined solution branches exhibiting "slanted snaking" [3]. Similar solutions with no-slip boundary conditions are no longer embedded in a background shear and exhibit standard snaking, i.e. snaking without a slant. These solutions may be stable [4]. Homotopic continuation from free-slip to no-slip boundary conditions is used to track the changes in the properties of the solutions and the associated bifurcation diagrams. An explanation of the results is given.

[1] Veronis, J. Fluid Mech. 5, 401–435 (1959)
[2] Beaume et al., J. Fluid Mech. 717, 417–448 (2013)
[3] Dawes, SIAM J. Appl. Dyn. Syst. 7, 186–206 (2008)
[4] Beaume et al., submitted

8:13AM M20.00002 Electro-convective instability at an ion-selective membrane , MATHIAS B. ANDERSEN, CLARA L. DRUZGALSKI, JOSEPH W. NICHOLS, ALI MANI, Center for Turbulence Research, Stanford University — In electrochemical engineering processes a major unresolved problem is the theoretical understanding of transport above the nominal diffusion limitation. When an electric current is passed from an aqueous electrolyte into an ion-selective membrane, ionic depletion next to the surface leads to transport limitation for a stagnant electrolyte. However, it has been shown that electrolytes under such conditions are hydrodynamically unstable when biased above a critical voltage. Mixing by the resulting flow can lead to enhanced transport. In this presentation we touch upon different elements of two studies of electro-convective instability at an ion-selective membrane: (1) the linear spatiotemporal stability when subject to a plane-Poiseuille cross flow, and (2) the chaotic transport characteristics at high voltages (cross flow absent). In (1) we identify absolutely and convectively unstable regimes and show that the imposed shear acts as a stabilization mechanism.

8:26AM M20.00003 One-way Euler equations: a novel spatial marching technique for convective instabilities , AARON TOWNE, TIM COLONIUS, California Institute of Technology — The parabolized stability equations (PSE) are a tool for rapid computation of convectively unstable flows. The efficiency of the method is achieved by solving the equations in frequency space using a spatial marching technique in the downstream direction. Unfortunately, the PSE operator contains upstream propagating acoustic modes that cause instability in this march unless these waves are numerically damped. Existing damping techniques introduce additional error into the solution and in particular contaminate the acoustic mid- and far-field. We have developed a method that removes the upstream acoustic modes from the linearized Euler equations without damaging the downstream modes. The upstream and downstream dynamics are decoupled using a recursive filtering technique that was originally developed for generating non-reflecting boundary conditions. The decoupled downstream modes are then evolved in the downstream direction. Our talk will focus on the validation of this method through comparison with PSE and direct solutions of the linearized Euler equations.

8:39AM M20.00004 Doubly-shocked Richtmyer-Meshkov Instability , VARAD KARKHANIS, PRAVEEN RAMAPRABHU, University of North Carolina at Charlotte — We report on detailed numerical simulations of a doubly-shocked Richtmyer-Meshkov Instability where two successive incident shock waves interact with a sinusoidally perturbed material interface. The problem is relevant to Inertial Confinement Fusion, type IA supernovae, and the design of mix experiments where multiple incident shocks have been proposed to potentially achieve freeze-out. In our simulations, the timing of the second incident shock was varied to realize a finite-amplitude initialization of the RM instability. The simulations were performed at two Atwood numbers, A = 0.15 and A = -0.99, where the latter condition is relevant to ejecta formation. For A = -0.99, the shock-interface interactions result in two successive phase inversions corresponding to the passage of the shocks from heavy to light media in each instance. We have investigated initial interface perturbations of different forms including sinusoidal, triangular and sawtooth waveforms and compare the growth rates from each interaction with linear and nonlinear models [1,2].

[1] Guy Dimonte and P. Ramaprabhu, Phys. Fluids 22, 014104 (2010).
[2] Guy Dimonte, Guillermo Terrones, F.J. Cherne and P. Ramaprabhu, J. Appl. Phys. 113, 024905 (2013).

8:52AM M20.00005 Numerical Simulations of the Single-mode, Reacting Richtmyer-Meshkov Instability Using Detailed Chemistry , NITESH ATTAL, PRAVEEN RAMAPRABHU, University of North Carolina at Charlotte — The interaction of a shock wave with a chemically reacting front is of importance to the design of supersonic combustors and scramjets where mixing from the Richtmyer-Meshkov Instability (RMI) could be tapped to increase combustion efficiency. We will describe results of shock-driven, reacting RMI of a sinusoidally perturbed, single-mode interface separating Hydrogen (fuel) and Oxygen at 300K and 1625K respectively. The non-premixed interface was accelerated by a Mach 1.2 shock traversing from the light (H_2) to heavy (O_2) fluid (Atwood number = 0.5) in a numerical shock tube of aspect ratio 12. The 2D simulations were performed using the compressible flow code FLASH [1], with modifications [2] to handle detailed chemistry and temperature-dependent material properties. The initial thickness of the material interface was systematically varied to study the effect of the diffusion thickness on the flame and instability dynamics. Product formation and heat release as a result of chemical reactions were described according to the 9-species, 19-steps detailed reaction mechanism [3].

[1] B. Fryxell et al., Astrophys. J., Suppl. Ser. 131, 273 (2000)
[2] N. Attal et al., Comput. Fluids (submitted for review)
[3] G. Billet, J. Comput. Phys. 204, 319 (2005)

9:05AM M20.00006 Behavior of embedded phase in shock-driven two-phase flow[1] , GARRETT KUEHNER, PATRICK WAYNE, DELL OLMSTEAD, CLINT CORBIN, TENNILLE BERNARD, PETER VOROBIEFF, C. RANDALL TRUMAN, The University of New Mexico — We present an experimental study of droplet acceleration in a shock-driven two-phase flow (air with embedded liquid droplets). The droplets (propylene glycol, diameter 0.5-3 μm) were pre-mixed with the air in the test section of a shock tube, then impulsively accelerated with planar shock wave with a Mach number of 1.7. A cross-section of the flow is illuminated with multiple pulses from Nd:YAG lasers, producing time-resolved visualizations of the seeded volume. The images are then analyzed to quantify droplet velocity and acceleration from the shock passage to about 1.5 ms after the shock. Based on the velocity measurements, we can resolve the droplet lag after the shock, when the massive droplets "catch up" with the flow of the surrounding air, as well as validate our earlier estimates of boundary layer growth.

[1]This research is supported by NNSA (US National Nuclear Security Agency).

9:18AM M20.00007 Multicomponent Reynolds-Averaged Navier–Stokes Simulations of Reshocked Richtmyer–Meshkov Instability and Turbulent Mixing: Reshock Time and Atwood Number Effects[1] , TIBERIUS MORAN-LOPEZ, National Nuclear Security Administration, OLEG SCHILLING, Lawrence Livermore National Laboratory — Reshocked Richtmyer–Meshkov turbulent mixing of gases with various Atwood numbers and shock Mach numbers is simulated using a third-order weighted essentially nonoscillatory implementation of a K–ϵ multicomponent Reynolds-averaged Navier–Stokes model. First, mixing layer widths from simulations with Mach number $Ma = 1.20$, Atwood number $At = 0.67$ (air/SF_6), and different times of reshock are shown to be in very good agreement with the experimental data of Leinov et al. [J. Fluid Mech. **626**, 449 (2009)]. Second, widths from simulations with $Ma = 1.50$ and $At = \pm 0.21$, ± 0.67 and ± 0.87 (corresponding to air/CO_2, air/SF_6 and H_2/air) are compared to the large-eddy simulation data of Lombardini et al. [J. Fluid Mech. **670**, 439 (2011)] and discussed. Budgets of the turbulent transport equations are considered to elucidate the mechanisms contributing to turbulent mixing in reshocked Richtmyer–Meshkov instability. Convergence of the mixing layer widths, mean fields, and turbulent fields under grid refinement is also assessed.

[1]This work was funded by the U. S. Department of Energy NNSA under the Predictive Science Academic Alliances Program by grant DE-FC52-08NA28616 and performed under the auspices of the DOE by LLNL under Contract DE-AC52-07NA27344.

9:31AM M20.00008 Linear and Nonlinear Simulations of the Richtmyer- Meshkov Instability in Magnetohydrodynamics[1] , RAVI SAMTANEY, ABEER BAKSH, SONG GAO, King Abdullah University of Science and Technology, VINCENT WHEATLEY, University of Queensland — Nonlinear ideal magnetohydrodynamics (MHD) simulations and analysis indicate that the Richtmyer-Meshkov instability (RMI) is suppressed in the presence of a magnetic field in Cartesian slab geometry. We present results of linear and nonlinear MHD simulations of RMI in cylindrical geometry. The linear simulations are performed with a numerical method that is an extension of the method proposed by Samtaney (J. Comput. Phys. 2009). In the absence of a magnetic field, linear analysis indicates that RMI growth rate during the early time period is similar to that observed in Cartesian geometry. However, this RMI phase is short-lived and followed by a Rayleigh-Taylor growth phase with an accompanied exponential increase in the perturbation amplitude. We examine several strengths of the magnetic field (characterized by $\beta = 2p/B^2$). For the strongest field case studied ($\beta \approx 2$) we see a significant suppression of the instability. We will present a description of the numerical methods, a complete characterization of the RMI linear stability in cylindrical geometry, and comparisons between linear and nonlinear MHD simulations for field strengths, and azimuthal and axial wavenumbers.

[1]RS and VW acknowledge partial support under an ARC Grant. AB, SG and RS are supported by KAUST.

9:44AM M20.00009 ABSTRACT WITHDRAWN –

9:57AM M20.00010 Instability of laterally heated cylindrical convection[1] , DE-JUN SUN, BO-FU WANG, University of Science and Technology of China — The three dimensional instabilities of axisymmetric flow in a vertical cylinder partially heated from the sidewall are explored. The cylindrical wall is heated in a central zone and is insulated above and below this zone, while both ends of the cylinder are cooled. The length of the heated zone equals to the cylinder radius. The dependence of the critical Rayleigh number on the Prandtl number is obtained for three fixed values of aspect ratio, A=1.92, 2, 2.1 (A=height/radius). The Prandtl number ranges from 0.02 to 6.7. The instability curve for A=1.92 is monotonous. The instability curves for A=2 and A=2.1 are non-monotonous and contain hysteresis, particularly, an instability island is found for A=2. The flow is oscillatory unstable at small Prandtl number due to hydrodynamic instability. At medium Prandtl number, the interaction of buoyancy and shear of base flow lead to the instability results. The Rayleigh-Benard instability is dominant at large Prandtl number, and the flow loses stability through a steady bifurcation.

[1]Supported by the National Natural Science Foundation of China under the Grant No. 11232011

Tuesday, November 26, 2013 8:00AM - 10:10AM –

Session M21 Biofluids: Locomotion IX - Bacteria and Microswimmers I 316 - Henry Fu, University of Nevada - Reno

8:00AM M21.00001 Induced Diffusion of Tracers in a Bacterial Suspension: Theory and Experiments , RODRIGO SOTO, Universidad de Chile, GASTON MINO, CNRS-ESPCI-Universite Paris 6 and Paris 7, JOCELYN DUNSTAN, University of Cambridge, ERIC CLEMENT, ANNIE ROUSSELET, CNRS-ESPCI-Universite Paris 6 and Paris 7 — The induced diffusion of tracers in a bacterial suspension is studied at low bacterial concentrations. Considering the swimmer-tracer hydrodynamic interactions at low-Reynolds number and using a kinetic theory approach, it is shown that the induced diffusion coefficient is proportional to the swimmer concentration, their mean velocity and a coefficient β. The coefficient scales as the tracer-swimmer cross section times the mean square displacement produced by single scatterings. Considering simple swimmer models it is shown that β increases for decreasing swimming efficiencies. Close to solid surfaces the swimming efficiency degrades and, consequently, the induced diffusion increases. Experiments on W wild-type *Escherichia coli* in a Hele-Shaw cell, under buoyant conditions, are performed to measure the induced diffusion on tracers near surfaces. The modification of the suspension pH varies the swimmers' velocity in a wide range allowing to extract the β coefficient with precision. It is found that that the solid surfaces modify the induced diffusion: decreasing the confinement height of the cell, β increases by a factor 4. The theoretical model reproduces this increase although there are quantitative differences, due to the simplified model.

8:13AM M21.00002 Rheological and boundary effects on microswimmers , THOMAS MONTENEGRO-JOHNSON, DANIEL LOGHIN, DAVID SMITH, University of Birmingham — Two important environmental factors impacting cell motility are the rheological properties of the surrounding fluid and the presence of boundaries. In this talk we will present simulations that explore the relationship between microswimmer, fluid rheology and boundary features, with a particular emphasis on the example of human sperm. Human sperm must navigate the labyrinthine structure of human fallopian tubes, actively bending their flagella in order to propel themselves through physiological mucus. Sperm trajectories are greatly affected by boundaries, scattering over features such as steps and ripples. We present simulations of scattering sperm-like swimmers in confined geometries, comparing these results to experiments of swimmers in microchannels. The rheological properties of mucus also affect sperms' ability to penetrate. Using the method of femlets, a new finite element technique entailing an immersed force representation of the swimmer with a body-fitted mesh, we present novel physical mechanisms through which shear-thinning, an important property of physiological mucus affects microscopic swimmers. In particular, we show that these effects are sensitive to the swimming stroke employed, and present example reciprocal swimmers that violate Purcell's Scallop Theorem.

8:26AM M21.00003 ABSTRACT WITHDRAWN –

8:39AM M21.00004 The signatures of microstructure in swimming properties of microorganisms in heterogeneous media , MEHDI JABBARZADEH, HENRY FU, University of Nevada, Reno — Many swimming microorganisms move through complex bioenvironments. Some of these environments, such as mucuses, contain a network of filaments with features at lengthscales comparable to the swimmers. In order to understand the effects of these heterogeneous microstructures on the swimming velocity of microorganism, we use Higdon's Slender Body Theory to study the hydrodynamic interaction between a Golestanian three-sphere swimmer and filaments. We find that in spatially varying background flows, there is an optimal length for the filament segments in Higdon's Slender Body Theory, and that the effect on swimming velocity is well-approximated by the Stokeslet contribution to the flow. We consider the effect of media composed of many such filaments. For isotropic media, the average change of the swimming velocity is determined by the density of the medium, while the variance of the swimming velocity depends on the filament structure through its density-density correlation function. The dependence of the variance on the medium can be understood by relating lengthscales of the medium microstructure to lengthscales of the swimmer's velocity field.

8:52AM M21.00005 Nutrient uptake in a suspension of squirmers , SHUNSUKE KAJIKI, YOHSUKE IMAI, TAKAMI YAMAGUCHI, TAKUJI ISHIKAWA, Tohoku University — Although microorganisms exist everywhere and significantly influence our life, little is known about the mass transport in suspensions of microorganisms. The aim of this study is to analyze the effect of the swimming motion on the nutrient uptake. In this study, we propose a discrete model of concentration field of nutrients in a microbial suspension, and simulate the nutrient uptake by model microorganisms. We modeled a microorganism as a squirmer, which swims by generating the tangential velocity on its surface. The hydrodynamic interactions between squirmers were calculated by Stokesian dynamics method. We first analyzed the uptake ratio of a squirmer in an infinite domain without any background flow, which agreed well with former study by Magar et al.(2003). Then, we investigated nutrient uptake process in an infinite suspension of squirmers with the volume ratio of 0.01-0.35. The results showed that the suspension uptake rate was strongly dependent on the volume ratio of squirmers, the swimming mode of squirmers and Peclet number. These results are important in understanding the transport phenomena in a microbial suspension.

9:05AM M21.00006 Hydrodynamic interaction of bacterial flagella – flagellar bundling[1] , SOOKKYUNG LIM, University of Cincinnati — Flagellar bundling is an important aspect of locomotion in bacteria such as Escherichia coli. To study the hydrodynamic behavior of helical flagella, we present a computational model that is based on the geometry of the bacterial flagellar filament at the micrometer scale. We consider two model flagella, each of which has a rotary motor at its base with the rotation rate of the motor set at 100 Hz. Bundling occurs when both flagella are left-handed helices turning counterclockwise (when viewed from the nonmotor end of the flagellum looking back toward the motor) or when both flagella are right-handed helices turning clockwise. Helical flagella of the other combinations of handedness and rotation direction do not bundle. In this work we use the generalized immersed boundary method combined with the unconstrained Kirchhoff rod theory, which allows us to study the complicated hydrodynamics of flagellar behavior. This is a joint work with Charlie Peskin at NYU.

[1]NSF

9:18AM M21.00007 Hydrodynamic model of bacterial tumbling near a non-slip surface[1] , JIAN SHENG, MEHDI MOLAEI, Texas Tech University — To swim forward, wild type *Escherichia coli* bacteria rotate their helical flagella CCW to form a bundle; to tumble, one or more flagella rotate CW to initiate flagella unbundling and polymorphic transformation that leads to a significant change in cell orientation in comparison to original swimming direction. These random change of direction increases bacterial dispersion and also is long speculated to be a mechanism for perichtricous bacteria to escape from a surface. Our recent experimental results show that the tumbling frequency is substantially suppressed near a solid surface by 50%, and the bacterium tends to start a new run in the direction parallel to the surface. This suppression occurs at two cell length (including flagella) away from the surface whereby steric hindrance plays less significant role. Here we propose an analytical model based on hydrodynamic interaction between flagella and the solid surface. We utilize Slender Body Theory combined with the image system of the singularities for the Stoke-flow to quantify the flow around the bacterial flagella in the presence of a no-slip surface. The model includes two non-identical rigid helical flagella representing a bundle and single flagellum. We have showed that in the bulk, a repulsive force among flagella initiates the unbundling and consequently tumbling; however, in presence of a solid surface, the force is strongly mitigated that stabilize the bundle and suppress the tumbling.

[1]NIH, NSF, GoMRI

9:31AM M21.00008 Swimming of a Ciliated Microorganism , HANLIANG GUO, EVA KANSO, University of Southern California — We propose a 2D model to consider the locomotion of a ciliated microorganism in a viscous fluid. The model consists of a circular body whose boundary is covered by a finite number of cilia. Stokes paradox does not hold due to the self-propelling nature of the organism. Using a regularized Stokeslet method, we determine numerically the time-dependent swimming motion for prescribed kinematics (undulatory beat) of the individual cilium. Phase differences between neighboring cilia result in metachronal waves characteristic of biological cilia. We compare our results based on the discrete cilia approach with the envelope model proposed by JR Blake. We then study the net locomotion as function of the metachronal wave. We find that, for a given geometry and cilia density, there is an optimal wave number (phase difference) for locomotion in terms of velocity of propulsion and efficiency.

9:44AM M21.00009 Collective Swimming in a Suspension of Ellipsoidal Squirmers , KOHEI KYOYA, DAIKI MATSUNAGA, YOHSUKE IMAI, TAKAMI YAMAGUCHI, TAKUJI ISHIKAWA, Tohoku University — Some recent research efforts have demonstrated the importance of biomechanics in understanding certain aspects of microorganism behaviors such as locomotion and collective motions of cells. However, former studies had problems in accurately computing many-body interaction of model microorganisms. In this study, we propose a boundary element method, based on the double-layer representation, for calculating interactions of many-body swimmers in Stokes flow regime. The proposed method allows us to analyze a large system size that could not be handled before. The model microorganism is assumed to be ellipsoid and propels itself by generating tangential velocities on its surface. Two types of microorganisms were modeled by varying the surface velocity; one is a "puller" which has the thrust-generating apparatus in front of the body such as *Chlamydomonas*, and the other is a "pusher" which has the thrust behind the body such as bacteria or spermatozoa. We then analyze interactions of 100 pullers or pushers. In both cases, some sorts of collective swimming were observed. In particular, pullers and neutral swimmers created large clusters and generated coherent structures.

9:57AM M21.00010 Contribution of cell body to the thrust production of flagellate bacteria , BIN LIU, School of Engineering, Brown University, THOMAS R. POWERS, School of Engineering and Department of Physics, Brown University, KENNETH S. BREUER, School of Engineering, Brown University — We trace individual motile microorganisms using a digital 3D tracking microscope in which the microscope stage follows the motion of the target. Using this technology, we not only trace a single cell over extended duration but also obtain the cell kinematics with high spatial and temporal resolution. We apply this tracking microscope to a study of Caulobacter crescentus, a bacterium that moves up to 100 microns (or 50 body lengths) per second and reverses its direction of motion by switching the rotation direction of its single helical flagellum. We show that when the cell reverses the rotation direction of the right-handed flagellum, e.g., switching from CW (a pusher) to CCW (a puller), its cell-kinematics is not completely reversible. In case of a puller, the cell almost spins along its long axis. However, in case of a pusher, besides spinning, the cell body precesses along its swimming direction, following a helical trajectory. These two types of cell-kinematics contribute to different cell motilities: the pusher rotates slower for the same swimming speed. We present a resistive force theory to account for this behavior, and by computing the torque on the cell body, we show that the finite precession angle of the bacterial pusher is optimized for swimming with fixed torque.

Tuesday, November 26, 2013 8:00AM - 10:10AM –

Session M22 General Fluid Dynamics III 317 - Shahram Pouya, Michigan State University

8:00AM M22.00001 Flow interaction between multiple cross-flow inlets in a horizontal pipe or channel[1] , PRANAB N. JHA, Dept. of Mechanical Engineering, University of Houston, CHUCK SMITH, Apache Corporation, Houston, RALPH W. METCALFE, Dept. of Mechanical Engineering, University of Houston — Incompressible flow in horizontal channels and pipes with multiple cross-flow inlets was studied numerically. Flow interference among the inlets was studied using an axisymmetric pipe flow model with five cross-flow inlets. Three basic flow regimes - trickle flow, partially blocked flow and fully blocked flow - were identified with respect to the blocking of upstream inlets by the downstream ones. The effects of inlet pressure and inlet size on the flow regimes under steady state conditions were studied. The presence of these regimes was supported by field data obtained from a horizontal natural gas well at two different times in the production cycle. Using a hydrostatic pressure model of reservoirs as the inlet boundary condition that drained fluid into the channel, the dynamic interaction of the inlets was studied. The transient behavior of the flow regimes was simulated and the key time-scales involved were identified. This is supported by field data where a similar behavior can be observed over time. Initially, the upstream inlets were in a blocked state, but opened up at a later time, leading to a trickle flow regime.

[1]Supported in Part by Apache Corporation

8:13AM M22.00002 Direct numerical simulation of electrokinetic chaos driven by ion concentration polarization next to an ion-selective membrane[1] , CLARA DRUZGALSKI, MATHIAS ANDERSEN, ALI MANI, Stanford University — We present a three-dimensional direct numerical simulation (DNS) of electrokinetic instability and hydrodynamic chaos near an ion-selective membrane subject to normal electric current. We employ a non-dissipative computational algorithm to numerically solve the full Poisson-Nernst-Planck and Navier-Stokes equations for a symmetric binary electrolyte in a gap between an ion-selective surface and a stationary reservoir. In our simulation the numerically stiff electric double layer (EDL) and extended space charge (ESC) regions are resolved without resorting to asymptotic models. Our 3D results enable visualization of the onset of instability and detailed flow structures over a wide range of scales and distances to the membrane. We present the time-averaged statistics from this simulation and its comparison to an otherwise identical 2D calculation. These simulations demonstrate transport phenomena that are crucial for accurate prediction of charge and mass transport in various electrochemical systems such as flow batteries and electrodialysis and electrolysis cells.

[1]Direct numerical simulation of electrokinetic chaos driven by ion concentration polarization next to an ion-selective membrane

8:26AM M22.00003 Mixing Dynamics Between Water and Biofuels , ALINE COTEL, AVERY DEMOND, JIARIU LEI, ERICA GREEN, University of Michigan — Currently, ethanol-based biofuels are considered to be among the best alternatives to gasoline. However, the potential environmental impact of a spill of such fuels on aquatic environments is an area of open discussion and research. Since these fuels are a combination of a miscible fluid (ethanol) and an immiscible fluid (gasoline), models used for traditional gasoline fuels (immiscible in water) are not applicable. Preliminary experiments show that when a solution of ethanol and glycol is mixed with water, a third mixed fluid is formed. Two distinct mixing regimes are observed. A turbulent wake is created between the ethanol/glycol and water layers to cause the ethanol and glycol solution to entrain and mix into with the water phase. In the first regime, due to nonlinear mixing behavior, a dramatic overturning is possible for a certain range of parameters. The second regime begins when the turbulent wake has dissipated and the internal wave created by the plate has begun to settle, typically within the first minute. At this point, Bénard-like cells, similar to those typically seen in Rayleigh-Bénard convection, form at the interface and relatively slow mass transfer is evident. Both regimes are described quantitatively with a set of dimensionless parameters.

8:39AM M22.00004 Measurement of Submerged Oil/Gas Leaks using ROV Video , FRANKLIN SHAFFER, USDOE National Energy Technology Laboratory, GIORGIO DE VERA, KENNETH LEE, ÖMER SAVAS, U.C. Berkeley Mechanical Engineering Dept — Drilling for oil or gas in the Gulf of Mexico is increasing rapidly at depths up to three miles. The National Commission on the Deepwater Horizon Oil Leak concluded that inaccurate estimates of the leak rate from the Deepwater Horizon caused an inadequate response and attempts to cap the leak to fail. The first response to a submerged oil/gas leak will be to send a Remotely Operated Vehicle (ROV) down to view the leak. During the response to the Deepwater Horizon crisis, the authors Savas and Shaffer were members of the Flow Rate Technical Group's Plume Team who used ROV video to develop the FRTG's first official estimates of the oil leak rate. Savas and Shaffer developed an approach using the larger, faster jet features (e.g., turbulent eddies, vortices, entrained particles) in the near-field developing zone to measure discharge rates. The authors have since used the Berkeley Tow Tank to test this approach on submerged dye-colored water jets and compressed air jets. Image Correlation Velocimetry has been applied to measure the velocity of visible features. Results from tests in the Berkeley Tow Tank and submerged oil jets in the OHMSETT facility will be presented.

8:52AM M22.00005 ABSTRACT WITHDRAWN –

9:05AM M22.00006 Aerodynamic damping of oscillating cantilevers from side walls in close proximity , ANDREW EASTMAN, MARK KIMBER, University of Pittsburgh — As a result of their simplicity, low power consumption, and relative ease of implementation, oscillating cantilevers have been investigated for use in many applications. However, use in many circumstances requires a close proximity to one or multiple boundaries. This can cause added damping to the flow that inhibits the operational effectiveness. This paper investigates the fluidic damping effect of two boundaries parallel to the oscillating motion of the fan at multiple voltage and frequency inputs. Experiments performed across a range of operating conditions showed that decreasing the distance between the boundaries and the oscillating cantilever increases the aerodynamic damping, which can be as high as 5X compared to normal (i.e., without sidewalls) operation. However this also decreases the power consumption which can be beneficial under certain circumstances. The power consumption also peaks when operating at resonance. The findings in the paper are significant to creating a basis to judge how to best operate an oscillating cantilever to achieve the maximum operational effectiveness.

9:18AM M22.00007 Growth and Decay of Fully-Developed Dean Flow , JESSE AULT, JOHN DAVIS, Princeton University — Flows in curved pipes are ubiquitous in piping systems, hydraulics, and even biological systems. Beginning with the pioneering work of Dean in the 1920s, there have been many studies of the development of flow in curved pipes. However, the opposite problem has been little studied: the decay of fully-developed flow in a curved pipe after exiting into a straight outlet. First, we study the entry length of flow in a curved pipe and compare our results with previously published work. Then, we study the problem of decay and provide a description of the transition of the fully-developed curved-pipe flow to the fully developed flow in a straight pipe. In particular, OpenFOAM's icoFOAM solver is used to perform direct numerical simulations in the geometry of curved pipes of varying curvature connected with straight outlets. The magnitude of vorticity, wall shear stress, and centerline velocity are plotted along the pipe while varying the Reynolds number and the ratio of the pipe radius to the radius of curvature. The development and decay lengths of the curved-pipe flow are then related to these two parameters. A simple model for the observed dynamics will be presented.

9:31AM M22.00008 An analytical approach to fluid ratcheting in oscillatory boundary layer[1] , JIE YU, Civil, Construction and Environmental Engineering, North Carolina State University — It is well known that oscillatory flows close to a rigid or flexible boundary induces a steady streaming due to viscosity. Under progressive motions, this becomes a unidirectional streaming near the boundary (e.g. mass transport or peristaltic pumping in water waves). This mechanism is shared by the phenomenon of ratcheting fluid in a narrow channel by vibrating the channel walls that are lined with asymmetric corrugations (shown by a recent experiment BAPS.2010.DFD.HC.3). A theory is presented here to describe the ratcheting effects in such a channel. A conformal transformation method, developed for waves over arbitrary periodic topographies (Yu & Howard, *J. Fluid Mech.* 2012), is adapted to deal with large corrugations of the channel walls. Under the assumption that the wall oscillations are of small amplitude, the vorticity dynamics can be analyzed in the mapped plane, obtaining the solution that describes the steady streaming field due to nonlinear convective inertia. The results are discussed, regarding the dependency of the pumping direction on the oscillation frequency of the walls and the effects of the end position relative to the phase of corrugations in the case of a finite length channel. Preliminary experimental data will be presented if time permits.

[1]Support by NFS (Grant CBET-0845957) during the period of this work is gratefully acknowledged.

9:44AM M22.00009 Multiple shock-induced luminescence in water , PEDRO QUINTO-SU, Instituto de Ciencias Nucleares, Universidad Nacional Autónoma de México, Apartado Postal 70-543, 04510, México D.F., México — Luminescence in microscopic volumes of water is observed as a result of the superposition of several laser-induced shocks created by optical breakdown.

9:57AM M22.00010 ABSTRACT WITHDRAWN –

Tuesday, November 26, 2013 8:00AM - 10:10AM –

Session M23 Geophysical: Atmospheric III 318 - Brad Marston, Brown University

8:00AM M23.00001 Statistically Steady-State Large Eddy Simulations of Subtropical Clouds With Time Varying Large Scale Forcing , KYLE PRESSEL, California Institute of Technology, TAPIO SCHNEIDER, ETH Zurich and California Institute of Technology, JOAO TEIXEIRA, Jet Propulsion Laboratory, California Institute of Technology, ZHIHONG TAN, California Institute of Technology — Despite the substantial effort directed towards understanding the role played by clouds in determining perturbed climate states, the combined effects of clouds remain a major, if not the most major, source of uncertainty in predictions of perturbed climate states. Here we discuss a new set of tools brought to bear on the cloud-climate problem, in particular statistically steady-state Large Eddy Simulations (LES) with dynamically consistent time-varying forcing provided by an atmospheric general circulation model (GCM). The simulations are used to investigate the response of subcloud-scale dynamics to variations in large scale dynamics for various perturbed climate scenarios. The results of this investigation are used to characterize the response of subtropical clouds to climate perturbations.

8:13AM M23.00002 Shear effects in the evaporatively driven cloud-top mixing layer , JUAN PEDRO MELLADO, Max Planck Institute for Meteorology — A stably stratified shear layer destabilized locally by moist convection is studied using direct numerical simulations as a model to investigate the role of evaporative cooling at the top of stratocumulus clouds in the presence of vertical mean shear. Velocity and time scales are obtained from the study of the vertical structure. It is found that, overlapping with the background shear layer that has been often documented in the cloud-free cases, with a thickness $(1/3)(\Delta u)^2/\Delta b$, where Δu and Δb are the velocity and buoyancy increments across the cloud top, the system develops a turbulence layer that is dominated by free convection inside the cloud and by shear production inside the relatively thin overlap region. As turbulence intensifies, the turbulence layer encroaches upwards into the background shear layer and defines thereby the entrainment velocity. This encroachment is well characterized by the penetration length formed with the in-cloud convective velocity and the buoyancy frequency inside the background shear layer. Consistently, the turbulence intensity inside the overlap region follows a mixed scaling combining the background mean shear and the in-cloud convective velocity.

8:26AM M23.00003 Sub-layers inside the entrainment zone of a dry, shear-free convective boundary layer[1] , JADE RACHELE GARCIA, JUAN PEDRO MELLADO, Max Planck Institute for Meteorology — The entrainment zone of a dry, shear-free convective boundary layer growing into a homogeneously stably-stratified fluid is studied using direct numerical simulation. Based on the self-similar analysis of the mean and variance buoyancy profiles, we identify two sub-layers within the entrainment zone, defined as the region of negative buoyancy flux: i) an upper sub-layer with a thickness comparable to the penetrative length scale based on the convective velocity and the buoyancy frequency of the free troposphere and ii) a lower sub-layer acting as a transition towards the mixed layer, with a thickness equal to a constant fraction of the boundary layer height. The capping region of the penetrative thermals belongs to the upper sub-layer of the entrainment zone, and the troughs between the penetrating thermals belong to the lower sub-layer of the entrainment zone. Correspondingly, different buoyancy scales are identified in the different regions; parametrizations thereof are provided and explained. This multiplicity of characteristic scales inside the entrainment zone helps to explain the uncertainty associated with previous analysis of entrainment zone properties and the difficulty to parametrize them based on a single length scale and a single buoyancy scale.

[1]Juelich Research Centre for the computing time

8:39AM M23.00004 Effect of systematic mode reduction on cloud formation and buoyancy transport in a model of moist turbulent convection[1] , JOERG SCHUMACHER, TU Ilmenau, THOMAS WEIDAUER, Dynardo GmbH Weimar — The insufficient parametrization of low clouds which are caused by shallow convection remains one of the biggest sources of uncertainty in large-scale models of global atmospheric motion. One way to overcome this lack of understanding is to develop simplified models of moist convection which allow for systematic studies of the cloud formation in different dynamical regimes. They provide an ideal testing bed for systematic and stepwise reductions of degrees of freedom. Such systematic reductions are studied here for a recently developed moist Rayleigh-Bénard convection model in the conditionally unstable regime. Our analysis is based on the Proper Orthogonal Decomposition (POD). The resulting reduced-order dynamical systems which are obtained by a projection of the original equations of motion onto the most energetic POD modes are found to reproduce important statistical quantities such as the cloud cover, liquid water fluxes and the global buoyancy transport to a very good degree. The number of modes can be compressed significantly before the POD models break down and cause significant deviations of essential mean transport quantities from the original fully resolved simulation data.

[1]The work was supported by the Deutsche Forschungsgemeinschaft.

8:52AM M23.00005 Multiscale Eddy Simulation for Moist Atmospheric Convection[1] , SAMUEL STECHMANN, University of Wisconsin-Madison, BJORN STEVENS, Max Planck Institute for Meteorology — A multiscale computational framework is designed for simulating atmospheric convection and clouds. In this multiscale framework, large eddy simulation (LES) is used to model the coarse scales of 100 m and larger, and a stochastic, one-dimensional turbulence (ODT) model is used to represent the fine scales of 100 m and smaller. Coupled and evolving together, these two components provide a multiscale eddy simulation (MES). Through its fine-scale turbulence and moist thermodynamics, MES allows coarse grid cells to be partially cloudy and to encompass cloud–clear air mixing on scales down to 1 m; in contrast, in typical LES such fine-scale processes are not represented or are parameterized using bulk deterministic closures. To illustrate MES and investigate its multiscale dynamics, a shallow cumulus cloud field is simulated. In comparison to LES, many statistical mean quantities are essentially the same in MES, which indicates that the bulk properties of the cloud fields are similar in LES and MES. However, MES has significantly larger turbulent kinetic energy and variance. To illustrate the fine-scale variability, an individual cloud is considered in detail, and partially cloudy grid cells are seen to be prominent near the cloud edges.

[1]Partially supported by grant NSF DMS 1209409.

9:05AM M23.00006 Direct numerical simulation of stationary homogeneous moist turbulence , DANIEL CHUNG, University of Melbourne, GEORGIOS MATHEOU, Jet Propulsion Laboratory California Institute of Technology — Direct numerical simulation is reported of stationary and homogeneous, buoyancy-driven turbulence in moist air. Moist-air dynamics is more complex than its dry-air counterpart because of the possibility of latent-heat release during condensation or latent-heat absorption during evaporation. These phase changes depend on the local fluid composition and alters the buoyancy in a non-trivial way. In this study, moist air is modeled using equilibrium thermodynamics and the continuum approach in which the effect of phase changes is manifested through a nonlinear dependence of buoyancy upon the mixture fraction. A large-scale forcing is imposed on the the mixture-fraction equation to model the engulfing action of large eddies. This flow represents an idealisation of subgrid-scale moist processes that occur in the simulation of clouds, and is a first step toward improving subgrid condensation schemes. Statistics from this flow, including cloud fraction, mean liquid water content and subgrid buoyancy flux, will be compared with the predictions of the commonly used Sommeria and Deardorff (1977) scheme.

9:18AM M23.00007 Laboratory study of orographic cloud-like flow , KANWAR NAIN SINGH, K.R. SREENIVAS, Jawaharlal Nehru Centre for Advanced Scientific Research, Bangalore — Clouds are one of the major sources of uncertainty in climate prediction, listed in "the most urgent scientific problems requiring attention" IPCC. Also, convective clouds are of utmost importance to study the dynamics of tropical meteorology and therefore, play a key role in understanding monsoons. The present work is to study the dynamics of orographic clouds. Parameterization of these clouds will help in forecasting the precipitation accurately. Also, one could validate laboratory results from our study by actually measuring cloud development along a sloping terrain. In this context a planar buoyant turbulent wall jet is considered as an appropriate low order fluid-dynamical model for studying the turbulence and entrainment in orographic-clouds. Flow is volumetrically heated to mimic the latent heat release due to condensation in an actual cloud. This is the first step in studying the entrainment dynamics of the evolving orographic cloud. We are going to present some results on the cloud development using techniques that allows us to construct a 3-dimensional flow field at each instance and its development over the time. By combining velocity field from PIV and flow volume from PLIF at successive instances, we estimate the entrainment coefficient. Since the life-cycle of a cloud is determined by the entrainment of ambient air, these results could be extremely helpful in understanding the dynamics of the clouds. Detailed results will be presented at the conference.

9:31AM M23.00008 Study of microphysical and radiative properties of contrail cirrus using large-eddy simulations , ROBERTO PAOLI, ODILE THOURON, DANIEL CARIOLLE, CERFACS — Contrails are ice clouds that form by condensation of water vapor exhaust from aircraft engines and develop further in the wake as they are entrained by the airplane trailing vortices. When contrails spread to form contrail cirrus, they can persist for hours resulting in additional (artificial) cloud cover that adds to the cover due to natural cirrus. This talk presents recent results from large-eddy situations (LES) of contrail cirrus dispersion that are carried out using the atmospheric model Méso-NH. The objective is to investigate whether and how the ambient conditions and the microphysical and optical properties of ice crystals (e.g. shape, albedo), affect the three-dimensional structure and the overall microphysical and radiative characteristics of the contrail. The analysis is carried out by changing the radiative properties of the atmosphere (e.g. day/night conditions) for a given level of atmospheric turbulence. The turbulent field is generated by means of a stochastic forcing technique that reproduces the atmospheric conditions encountered in the upper troposphere. In addition to helping understanding the physics of contrails, the LES data retrieved from this study may provide useful inputs to the parameterization of contrail cirrus into global or climate models.

9:44AM M23.00009 Large eddy simulations of Arctic mixed-phase clouds , COLLEEN M. KAUL, JOAO TEIXEIRA, GRAEME L. STEPHENS, Jet Propulsion Laboratory, California Institute of Technology — Mixed-phase stratocumuli have been observed to persist in the Arctic for hours or even days, despite the inherent instability of liquid droplet–ice particle mixtures. Since mixed-phase and ice-only clouds have very different radiative effects, identifying the factors that allow the maintenance of mixed-phase clouds is an important component of understanding Arctic climate. Various feedbacks between turbulence, radiation, and microphysical processes are hypothesized to exist, but further information about these conjectured feedback mechanisms is needed. Prior large eddy simulation studies of Arctic mixed-phase clouds have largely focused on the details of their microphysical modeling, although microphysical processes alone cannot explain the longevity of mixed-phase Arctic stratocumuli. Therefore, this study investigates the representation of turbulence in large eddy simulations of such clouds, considering the effects of turbulence closure, grid resolution, and domain size on the predicted cloud characteristics in three different case studies.

9:57AM M23.00010 Turbulent Mixing at the Edge of a Cloud , RAYMOND SHAW, MATTHEW BEALS, Michigan Technological University, JACOB FUGAL, Max Planck Institute for Chemistry, BIPIN KUMAR, Ilmenau University of Technology, JIANG LU, Michigan Technological University, JOERG SCHUMACHER, Ilmenau University of Technology, JEFFREY STITH, National Center for Atmospheric Research — Numerical and field experiments have been brought to bear on the question of how atmospheric clouds respond when they experience turbulent mixing with their environment. Simply put, we ask when a cloud is diluted, do all droplets evaporate uniformly (homogeneous mixing) or does a subset of droplets evaporate completely, leaving the remaining droplets unaffected (inhomogeneous mixing)? First, the entrainment of clear air and its subsequent mixing with a filament of cloudy air is studied in DNS that combine the Eulerian description of the turbulent velocity, temperature and vapor fields with a Lagrangian cloud droplet ensemble. The simulations provide guidance on the proper definition of the thermodynamic response time for the Damkoehler number, and demonstrate the transition from inhomogeneous to homogeneous mixing as mixing progresses within the inertial subrange. Second, an airborne digital holographic instrument (Holodec) shows that cloud edges are inhomogeneous at the centimeter scales. In local cloud volumes the droplet size distribution fluctuates strongly in number density but with a nearly unchanging mean droplet diameter, until the fluctuations finally cascade to the centimeter scale, when the droplet diameter begins to respond.

Tuesday, November 26, 2013 8:00AM - 10:10AM –

Session M24 Compressible Flows II: Experimental Methods 319 - Michael Hargather, New Mexico Institute of Mining and Technology

8:00AM M24.00001 IR thermography measurements on roughness induced transition. , FRANCESCO AVALLONE, University of Naples Federico II, FERRY F.J. SCHRIJER, Delft University of Technology, GENNARO CARDONE, University of Naples Federico II — The current investigation focusses on the effect of surface protrusions and indentations on the occurrence of transition. The objective of this work is to characterize the roughness-induced transition in the hypersonic Ludwieg tube of Delft University of Technology at Mach 7.5 and a unit Reynolds number ranging from $8 \cdot 10^6 m^{-1}$ to $14 \cdot 10^6 m^{-1}$. The tests are carried out on a 200 mm long, 5° compression ramp on which the 1 and 2 mm high tripping elements were positioned at 30 and 60 mm from the leading edge. This configuration resulted in a non-dimensional trip height k/δ ranging from 0.5 to 1 where k is the trip height and δ the laminar boundary layer thickness at the trip location. Measurements were performed for isolated and distributed three-dimensional roughness elements having various shapes (cylinder, square, swept ramp, half sphere) and dimensions. Additionally two dimensional cavities and three dimensional cavities are tested and their effectiveness is compared to the protruding elements. The flow field downstream of the roughness elements is analyzed by means of quantitative infrared themography and schlieren visualization. Finally some first results obtained using particle image velocimetry are presented.

8:13AM M24.00002 Supersonic Jet Mixing with Vibrational Non-Equilibrium[1] , HEATH H. REISING, UTSAV KC, PHILIP L. VARGHESE, NOEL T. CLEMENS, University of Texas at Austin — A new study has been initiated to study the effect of vibrational non-equilibrium on turbulent mixing and combustion. This work is relevant to high-speed, high-temperature environments, such as scramjet combustors, where shocks and mixing can lead to high degrees of vibrational non-equilibrium. In this experimental study, a new facility has been developed that consists of a perfectly-expanded axisymmetric Mach 1.5 turbulent air jet issuing into an electrically heated co-flow of air for precise control of the temperature and thus vibrationally-active population. This hot flow can be brought into non-equilibrium when the co-flow fluid is rapidly mixed with the colder supersonic jet fluid. Effects of the non-equilibrium can be isolated by replacing the nitrogen in the flow with argon. The degree of non-equilibrium in the jet shear layers is quantified by using high-spectral resolution time-averaged spontaneous Raman scattering centered on the Stokes-shifted Q branch line of N_2 at 607 nm. In this first phase of the study, the effect of non-equilibrium on the mixing field will be investigated, but future work will focus on H_2-air combustion. Planar Rayleigh thermometry is utilized to investigate the effects of vibrational non-equilibrium on the turbulent structures and thermal dissipation field.

[1]This work was funded by the Air Force Office of Scientific Research under BRI grant FA9550-12-0460.

8:26AM M24.00003 Resolving turbulence in hypersonic flows using PIV , OWEN WILLIAMS, Princeton University, TUE NGUYEN, Nanyang Technological University, ALEXANDER J. SMITS, Princeton University, Monash University — Measuring turbulence in hypersonic flow using PIV is made difficult by high dynamic range requirements and low flow density, which leads to stringent particle sizing requirements to avoid filtering the turbulent signal. Particle frequency response is usually measured using a strong shock and conventionally taken to be representative of the entire flow. A particle dynamics model is used to demonstrate how measured frequency responses depend on shock strength in hypersonic flow, due to changes in particle drag associated with finite inertia, compressibility and slip. A method is presented to extrapolate freestream shock responses to other positions in the flow with disturbance levels comparable to turbulent motions. This method leads to estimates of the variation in frequency response and Stokes number; highlighting regions of the flow which might suffer from filtering. We will demonstrate practical improvements to particle response and the resulting change in turbulent filtering.

8:39AM M24.00004 Background-Oriented Schlieren Characterization of Explosions , CYNTHIA ROMO, MICHAEL HARGATHER, New Mexico Tech — Characterizing the energy release from large explosions is a difficult process using traditional point-pressure gages. The background oriented schlieren technique is used here to provide large-field-of-view visualization of the shock wave propagation from large-scale explosions. This technique is used to allow field-measurements of blast wave properties instead of traditional point-wise measurements. By analyzing the shock wave propagation Mach number the peak overpressure and overpressure duration are estimated for different explosions. The technique is applied to the visualization of encased explosions, including car bombs, to estimate the amount of shock energy lost to the fragmentation and acceleration of the casing. Comparisons are made to un-encased explosions. The optically measured data is compared to experimental data recorded using piezoelectric pressure transducers. Scaling relationships are examined to determine scalability of encased explosions.

8:52AM M24.00005 Quantitative schlieren measurement of shock wave pressure profile , JESSE TOBIN, MICHAEL HARGATHER, New Mexico Tech — Quantitative schlieren imaging is used to measure the pressure profile of a shock wave in air. The quantitative schlieren technique uses a weak lens calibration object to relate pixel intensity values in schlieren images to a known refractive index gradient. The refractive index gradient is converted to a density gradient, which in turn is converted to a pressure distribution using an approximated local air temperature. A high-speed digital camera is used to record schlieren images of shock wave propagation. Post-processing of the image record determines the changes in pixel intensity, and thus the density and pressure distributions across the shock front. The calculated pressure profile is compared to measurements performed using a piezoelectric pressure transducer. The quantitative schlieren measurement approach is benchmarked using a laminar flat plate free-convection boundary layer.

9:05AM M24.00006 Characterization of Magnetohydrodynamic (MHD) Shock Sensor using Schlieren Imaging , OWEN ROCKWELL, MICHAEL HARGATHER, New Mexico Tech — Schlieren imaging is used to quantitatively determine the speed and pressure duration of a shock wave traveling through air. The high-speed quantitative schlieren images are then used to characterize a new magnetohydrodynamic (MHD) shock sensor. This device uses the air density and particle velocity changes across a shock wave to determine the shock velocity via the distortion of a magnetic field. Using Faraday's law of electromagnetic induction, the shock velocity and pressure can be interpreted from a change in potential across the electrodes within the device. This principle along with the assumption that the shock wave is traveling through the undisturbed air allows for the calculation of shock velocity. Piezoelectric pressure gauges are used for comparison to measure the pressure pulse magnitude and duration.

9:18AM M24.00007 Quantitative image processing of high-speed Schlieren of a hot supersonic jet[1] , TOBIAS ECKER, DONALD R. BROOKS, K. TODD LOWE, WING F. NG, Virginia Tech — Understanding the physics of noise generation from hot supersonic jets is indispensable in the effort of jet noise reduction. This study describes the analysis of time-resolved Schlieren images obtained in a hot supersonic jet with Mach wave radiation ("crackle"). Proper orthogonal decomposition (POD) is commonly used with large quantitative experimental and numerical datasets. Recent research has shown application of POD post-processing with flow visualization techniques in order to extract valuable information on the large-scale turbulent structures in the flow. POD of the intensity distributions of the Schlieren images were performed to reveal organized structures in the outer shear layer, while mode evolution may be reconstructed in the images taken at over 86,400 frames per second. Albeit the screeching modes are more than twice as energetic than the next order of modes for NPR = 3, TTR = 2.5, a number of modes of similar energy density were found to exhibit convective cyclical structures.

[1]The authors wish to acknowledge the Office of Naval Research for funding this work through the Jet Noise Reduction Project (ONR BRC: N00014-11-1-0754, ONR DURIP: N00014-12-1-0803), program manager Dr. Brenda Henderson.

9:31AM M24.00008 Turbulence measurements in high-speed flows using the Focusing Laser Differential Interferometer , MATTHEW FULGHUM, GARY SETTLES, Penn State University — The Focusing Laser Differential Interferometer (FLDI) was invented by Smeets at ISL in the 1970s, and was used recently by Parziale in the CalTech T5 shock tunnel. It is a relatively-simple, non-imaging common-path interferometer for measuring refractive signals from transition and turbulence, and it has a unique ability to look through facility windows, ignore sidewall boundary-layers and vibration, and concentrate only on the signal near a pair of sharp beam foci in the core flow. Benchtop experiments using a turbulent helium jet in air demonstrate focusing ability, frequency response, unwanted signal rejection, and ease of use. The FLDI is then used to measure freestream turbulence intensity and spectra in the PSU supersonic wind tunnel at Mach 3, with results compared to hot-wire-anemometer data. A special feature of the FLDI instrument used here is the replacement of traditional fixed Wollaston prisms with variable Sanderson prisms for laser-beam separation and recombination. Research sponsored by AEDC Hypervelocity Tunnel 9.

9:44AM M24.00009 Synthetic streak images (x-t diagrams) from high-speed digital video records , GARY SETTLES, Penn State University — Modern digital video cameras have entirely replaced the older photographic drum and rotating-mirror cameras for recording high-speed physics phenomena. They are superior in almost every regard except, at speeds approaching one million frames/s, sensor segmentation results in severely reduced frame size, especially height. However, if the principal direction of subject motion is arranged to be along the frame length, a simple Matlab code can extract a row of pixels from each frame and stack them to produce a pseudo-streak image or x-t diagram. Such a 2-D image can convey the essence of the large volume of information contained in a high-speed video sequence, and can be the basis for the extraction of quantitative velocity data. Examples include streak shadowgrams of explosions and gunshots, streak schlieren images of supersonic cavity-flow oscillations, and direct streak images of shock-wave motion in polyurea samples struck by gas-gun projectiles, from which the shock Hugoniot curve of the polymer is measured. This approach is especially useful, since commercial streak cameras remain very expensive and rooted in 20th-century technology.

9:57AM M24.00010 Plenoptic PIV: Towards simple, robust 3D flow measurements[1] , BRIAN THUROW, TIM FAHRINGER, Auburn University — In this work, we report on the recent development of plenoptic PIV for the measurement of 3D flow fields. Plenoptic PIV uses a plenoptic camera to record the 4D light-field generated by a volume of particles seeded into a flow field. Plenoptic cameras are primarily known for their ability to computational refocus or change the perspective of an image after it has been acquired. In this work, we use tomographic algorithms to reconstruct a 3D volume of the particle field and apply a cross-correlation algorithm to a pair of particle volumes to determine the 3D/3C velocity field. The primary advantage of plenoptic PIV over multi-camera techniques is that it only uses a single camera, which greatly reduces the cost and simplifies a typical experimental arrangement. In addition, plenoptic PIV is capable of making measurements over dimensions on the order of 100 mm x 100 mm x 100 mm. The spatial resolution and accuracy of the technique are presented along with examples of 3D velocity data acquired in turbulent boundary layers and supersonic jets.

[1]This work was primarily supported through an AFOSR grant.

Tuesday, November 26, 2013 8:00AM - 9:57AM –

Session M25 Flow Control VII: Actuator Design and Analysis 320 - David Ashpis, NASA

8:00AM M25.00001 Single Dielectric Barrier Discharge Plasma Actuator Modelling using a Charge Transport Approach , THEODORE WILLIAMS, THOMAS CORKE, University of Notre Dame — Single dielectric barrier discharge (SDBD) plasma actuators have been used in active flow control due to their benefits of high response rate, no mechanical parts, and low cost. To effectively model the aerodynamic effects of a SDBD using computational fluid dynamics, a numerically efficient model of SDBD plasma actuator parameters is required. This work presents a charge transport model that is able to simulate the dynamic characteristics of an AC plasma actuator and calculate the time-dependent body force vector distribution. This work improves upon previous models by being able to simulate high-curvature electrode surfaces. Validation of this work is performed by comparison to experimentally measured thrust values.

8:13AM M25.00002 SDBD Plasma Actuator and Geometric Optimization for Optimal Flow Control of Wind Turbine Blades , THOMAS CORKE, THEODORE WILLIAMS, ALEKSANDAR JEMCOV, JOHN COONEY, University of Notre Dame — A Quantitative Design Optimization approach for active flow control using SDBD plasma actuators is presented. The approach couples passive geometric changes and plasma actuator design to produce a "compliant flow" that maximizes control authority. Aerodynamic shape optimization tools employed in this study make use of the adjoint formulation of the Navier-Stokes equations for incompressible flows. These are solved to obtain shape derivatives that are used in a gradient optimization procedure to produce aerodynamic shapes that are flow-control compliant. Coupling of compliant geometries and flow control devices are able to provide dynamic lift control to wind turbine blades. The effect of the plasma actuator is included as a body force distribution in the flow governing equations. The optimization seeks designs that effectively utilize a SDBD plasma actuator and are aerodynamically compliant to realize increased energy production on wind turbine blades.

8:26AM M25.00003 Thrust Measurement of Dielectric Barrier Discharge (DBD) Plasma Actuators[1] , DAVID E. ASHPIS, NASA Glenn Research Center, MATTHEW C. LAUN, Sierra Lobo Inc. — DBD plasma actuators generate a wall-jet that can be used for active flow control. We used an analytical balance to measure the thrust generated by the actuator, it is a common metric of its performance without external flow. We found that the measured force is afflicted by several problems; it drifts in time, not always repeatable, is unstable, and depends on the manner the voltage is applied. We report results of investigations of these issues. Tests were conducted on an actuator constructed of 1/4 inch thick high-density polyethylene (HDPE) dielectric with 100 mm long offset electrodes, with applied voltages up to 48 kV p-p and frequencies from 32 Hz to 2.5 kHz, and pure Sine and Trapezoidal waveforms. The relative humidity was in the range of 51-55%, corresponding to moisture range of 10,500 to13,000 ppm mass. Force readings were up to 500 mg, (approximately 50 mN/m). We found that the measured force is the net of the positive thrust generated by the wall-jet and an "anti-thrust" acting in the opposite direction. We propose a correction procedure that yields the plasma-generated thrust. The correction is based on voltage-dependent anti-thrust measured in the low frequency range of 20-40 Hz. We found that adjacent objects in a test setup affect the measured thrust, and verified it by comparing experiments with and without a metal enclosure, grounded and ungrounded. Uncorrected thrust varied by up to approximately $\pm$100%, and the corrected thrust variations were up to approximately 30%.

[1]Supported by NASA's FAP/Aerospace Sciences Project.

8:39AM M25.00004 Efficiency of Flow Control Actuators , AVRAHAM SEIFERT, Tel Aviv University — Bluff body flow control is an important and extensively studied branch of flow control. Bluff bodies can be found everywhere and mitigating its massively separated flow and associated penalties are extremely important from efficiency, vibration and noise considerations. Two distinct classes exist: one which involves separation control, i.e., where the flow can be reattached, and the other; control of massively separated flows, where the near wake is to be controlled. This paper reviews the state-of-the-art in actuators technology with the aim of providing a common ground for comparison and wise technology choices. Real-world aspects as well as fundamental challenges are identified and discussed. Actuators are sometimes considered, in a somewhat simplistic manner, as nominally 2D where high aspect ratio openings are assumed to lead to 2D excitation. This is clearly not the case and 3D effects always eventually prevail. The presentation will also discuss approaches not only to acknowledge and attempt to understand but also utilize 3D effects for effective and efficient flow control.

8:52AM M25.00005 Numerical Simulation of Nanosecond Pulsed Dielectric Barrier Discharge Actuator for Flow Control , J.G. ZHENG, Temasek Laboratories, National University of Singapore, Z.J. ZHAO, Department of Mechanical Engineering, NUS, J. LI, Y.D. CUI, B.C. KHOO, Temasek Laboratories, NUS, TEMASEK LABORATORIES, NUS TEAM — Recently, nanosecond pulsed dielectric barrier discharge (DBD) actuator has emerged as a promising active flow control means. In this study, numerical simulation is carried out to investigate fluid dynamics induced by nanosecond pulsed gas discharge. Two types of so-called phenomenological approaches reported in the literature are employed to model effects of plasma discharge. In the first methodology, the plasma region over covered electrode is modelled as preheated and pressurized gas layer. The second method is based on a quasi-one-dimensional self-similar, local ionization kinetic model. The plasma models are then coupled with compressible Navier-Stokes equations governing the external flow. The two models are validated against experimental data obtained for flow field arising from single voltage pulse discharge in quiescent air and proved to be valid. The numerical method is then applied to study flow separation control over NACA0015 airfoil with the actuator placed on the leading edge of airfoil. The goal is to numerically reproduce the formation and development of complex vortex structures due to plasma actuator induced shock propagation through the airflow. Special interest is focused on how the generated vortex interacts with and suppresses the separated shear layer.

9:05AM M25.00006 Nonlinear model-order reduction for oscillator flows using POD-DEIM , MIGUEL FOSAS DE PANDO, PETER J. SCHMID, LadHyX, CNRS-Ecole Polytechnique, DENIS SIPP, ONERA/DAFE — The design of control laws for fluid systems often relies on the prediction given by a reduced-order model of the response of the flow to actuations. Model-order reduction techniques have successfully been applied to flows exhibiting linear behavior. However, in many cases of practical interest the effect of nonlinearities must be incorporated to assess the dynamics of the flow. In this work, we present an extension to the POD-DEIM technique introduced by Chaturantabut and Sorensen (2010) to derive reduced-order models from flow solvers with minimal development effort. This technique will be demonstrated on the compressible flow around a NACA0012 airfoil featuring limit-cycle oscillations. Attention will then be focused on the accuracy and the robustness of the POD-DEIM reduced-order model at off-design conditions, and its application to flow control.

9:18AM M25.00007 Structural Sensitivity for Estimating Actuator and Sensor Placement for Flow Control[1] , MAHESH NATARAJAN[2], JONATHAN FREUND[3], DANIEL BODONY[4], University of Illinois at Urbana-Champaign — A control strategy is developed for the modification of the growth rates of global modes. The method is based on an analysis of the structural sensitivity of the baseflow, which uses the forward and adjoint global modes of the steady baseflow to estimate effective locations of actuation. Linear feedback is used to modify the eigenstructure of the linearized system for reduction/stabilization of amplification rates using different control-feedback pairs. This procedure provides an assessment of the effectiveness of different modes of actuation and different quantities to sense. The method is demonstrated for the case of a separated boundary layer in a Mach 0.7 diffuser and the eigensystem senstivity to perturbation is evaluated for different cases. An error analysis of the predicted and computed eigenvalues as a function of the control amplitude establishes the limit of applicability of the linear description. Direct numerical simulations demonstrate the efficacy of a linear feedback controller based on mass injection with density feedback.

[1]Rolls Royce North America
[2]PhD student in Department of Aerospace Engineering
[3]Professor in Department of Mechanical Science and Engineering
[4]Associate Professor in Department of Aerospace Engineering

9:31AM M25.00008 Increasing Wind Turbine Power Generation Through Optimized Flow Control Design , JOHN COONEY, THEODORE WILLIAMS, THOMAS CORKE, University of Notre Dame — A practical, validated methodology is outlined for implementing flow control systems into wind turbine designs to maximize power generation. This approach involves determining optimal flow control strategies to minimize aerodynamic losses for horizontal axis wind turbines during Region II operation. A quantitative design optimization (QDO) process is completed for the wind turbine utilized in the Notre Dame Laboratory for Enhanced Wind Energy Research. QDO utilizes CFD simulations and shape optimization tools to maximize effectiveness of flow control. Here, only flow control schemes that could be retrofitted on the existing turbine were explored. The final geometry is discussed along with accompanying validations of the predicted performance from wind tunnel experiments at full-scale conditions. Field data from the wind energy laboratory is included.

9:44AM M25.00009 Closed-loop turbulence control with machine learning methods[1] , BERND R. NOACK, THOMAS DURIEZ, LAURENT CORDIER, Institute PPRIME, France, MARC SEGOND, MARKUS ABEL, Ambrosys GmbH, Germany, STEVEN BRUNTON, University of Washington, USA, MAREK MORZYNSKI, Poznan University of Technology, Poland, JEAN-CHARLES LAURENTIE, VLADIMIR PAREZANOVIC, JEAN-PAUL BONNET, Institute PPRIME, France — We propose a machine learning control strategy for arbitrary turbulent flow configurations with finite number of actuators and sensors. This method designs and optimizes closed-loop control laws automatically detecting and exploiting linear to strongly non-linear actuation mechanisms. Presented examples range from a simple analytical model to the TUCOROM mixing layer control demonstrator.

[1]Funding of the ANR Chair of Excellence TUCOROM, of the EC's Marie-Curie ITN program and of Ambrosys GmbH is acknowledged.

Tuesday, November 26, 2013 8:00AM - 9:44AM –

Session M26 Reacting Flows VIII: General 321 - Carlos Pantano, University of Illinois at Urbana-Champaign

8:00AM M26.00001 Direct numerical simulation of turbulent autoigniting flames , RAJAPANDIYAN ASAITHAMBI, KRISHNAN MAHESH, University of Minnesota — A density based method for DNS/LES of compressible chemically reacting flows is proposed with an explicit predictor step for advection and diffusion terms, and a semi-implicit corrector step for stiff chemical source terms. This segregated approach permits independent modification of the Navier-Stokes solver and the time integration algorithm for the chemical source term. The numerical details are briefly summarized and results from autoigniting non-premixed flames in vitiated coflow with different fuels are discussed. We perform a direct numerical simulation of a turbulent round hydrogen jet at a Reynolds number of $\sim$12,500 injected into coflowing hot air. Flow statistics and the physics of the flame ignition and stabilization will be discussed.

8:13AM M26.00002 Solution of variable-density edge flames by a homotopy method , KAI-PIN LIAO, MOSHE MATALON, CARLOS PANTANO, University of Illinois at Urbana-Champaign — The edge flame is a fundamental flame structure essential to the description of flame hole dynamics in turbulent nonpremixed combustion and the stabilization of lifted jet flames. The edge flame propagation velocity is a solution to a nonlinear eigenvalue problem based on the variable-density reactive Navier-Stokes equations. This problem is remarkably difficult to solve as a boundary-value problem due to the two-dimensionality of edge flames and the nonlinear nature of the equations. In this talk we present a novel algorithm to solve for the steady state solution of the system using a homotopy method that maps continuously the easy-to-find constant-density solution into the variable-density flow. The flow and the combustion fields are segregated within an outer Picard iteration embedding a Newton method, which is solved sequentially using GMRES with proper multigrid preconditioners. This efficient algorithm enables the parametric study of the effects of thermal expansion, differential diffusion, heat release, and strain rate on edge flame structure and propagation velocity for variable-density flows. Furthermore, a discussion of admissible boundary conditions for this problem will be presented.

8:26AM M26.00003 Log-Normality and Multifractal Analysis of Flame Surface Statistics , ABHISHEK SAHA, SWETAPROVO CHAUDHURI[1], CHUNG K. LAW, Princeton University — The turbulent flame surface is typically highly wrinkled and folded at a multitude of scales controlled by various flame properties. It is useful if the information contained in this complex geometry can be projected onto a simpler regular geometry for the use of spectral, wavelet or multifractal analyses. Here we investigate local flame surface statistics of turbulent flame expanding under constant pressure. First the statistics of local length ratio is experimentally obtained from high-speed Mie scattering images. For spherically expanding flame, length ratio on the measurement plane, at predefined equiangular sectors is defined as the ratio of the actual flame length to the length of a circular-arc of radius equal to the average radius of the flame. Assuming isotropic distribution of such flame segments we convolute suitable forms of the length-ratio probability distribution functions (*pdf*s) to arrive at corresponding area-ratio *pdf*s. Both the *pdf*s are found to be near log-normally distributed and shows self-similar behavior with increasing radius. Near log-normality and rather intermittent behavior of the flame-length ratio suggests similarity with dissipation rate quantities which stimulates multifractal analysis.

[1]Currently at Indian Institute of Science, India

8:39AM M26.00004 Reactive transport modeling of CO_2 inside a fractured rock: Implications of mass transfer and storage capacity , MOHAMMAD ALIZADEH NOMELI, AMIR RIAZ, University of Maryland — A numerical model of geochemical transport is developed to evaluate long term mineral trapping of CO_2 inside a fractured rock. The problem contains flow of CO_2 between finite plates that represents a single fracture in post-injection regime. This study investigates the impact of fractures on CO_2 transport and storage capacity. The effect of surface roughness is also investigated to predict the actual efficiency of mineral trapping of CO_2 for a long period of time. The model is composed of direct numerical simulation tools and algorithms for incompressible flow and conservative transport combined with kinetics of corresponding chemical reactions. For each time step, transport and reactions are solved by means of finite difference method using a sequential non-iterative approach. It is found that the simple fracture is filled at the inlet because concentrations of carbonate ions are greater (more saturated states).

8:52AM M26.00005 Rayleigh-Taylor Unstable Flames – Fast or Faster?[1] , ELIZABETH HICKS, Northwestern University — The speed of a Rayleigh-Taylor unstable, premixed flame could plausibly be influenced by both the Rayleigh-Taylor instability of the flame front and the turbulence generated by the flame itself. Both of these mechanisms stretch and wrinkle the flame front, increasing its surface area and speed. But which of these two processes is dominant? Is the flame speed better modeled by the Rayleigh-Taylor speed or the root-mean-square velocity of the turbulence? To address these questions, we will present the results from three-dimensional, direct numerical simulations of Rayleigh-Taylor unstable flames that generate moderately turbulent conditions. We will discuss the influence of the Rayleigh-Taylor instability and turbulence on the flame front and focus on cases for which the flame speed substantially exceeds the laminar flame speed.

[1]Funded by CIERA at Northwestern University

9:05AM M26.00006 Thermal convection and gyrokinetic effects in inductively-coupled plasma-based lenses[1] , MILAD MORTAZAVI, JAVIER URZAY, ALI MANI, Center for Turbulence Research, Stanford — The principle of operation of a plasma lens consists of tuning the electron-density field, or equivalently, the refractive-index distribution in an ionized gas environment. The use of larger and more powerful lenses with higher electron-density results in higher optical performance and resolution, but also leads to hydrodynamic instabilities and noticeable bulk motion in the plasma, which may be detrimental for its optical performance. In this investigation, the effects of thermal convection and mean gyrokinetic motion are analyzed on an inductively-coupled Argon-plasma lens. The analyses utilize theoretical and computational methods to identify relevant characteristic parameters and operating regimes of interest for the optimal use of the plasma lens.

[1]Supported by DARPA.

9:18AM M26.00007 A constitutive theory of reacting electrolyte mixtures[1] , MARTINA COSTA REIS, University of Campinas- UNICAMP, YONGQI WANG, Technische Universitaet Darmstadt, ADALBERTO BONO MAURIZIO SACCHI BASSI, University of Campinas- UNICAMP — A constitutive theory of reacting electrolyte mixtures is formulated. The intermolecular interactions among the constituents of the mixture are accounted for through additional freedom degrees to each constituent of the mixture. Balance equations for polar reacting continuum mixtures are accordingly formulated and a proper set of constitutive equations is derived with basis in the Müller-Liu formulation of the second law of thermodynamics. Moreover, the non-equilibrium and equilibrium responses of the reacting mixture are investigated in detail by emphasizing the inner and reactive structures of the medium. From the balance laws and constitutive relations, the effects of molecular structure of constituents upon the fluid flow are studied. It is also demonstrated that the local thermodynamic equilibrium state can be reached without imposing that the set of independent constitutive variables is time independent, neither spatially homogeneous nor null. The resulting constitutive relations presented throughout this work are of relevance to many practical applications, such as swelling of clays, developing of bio and polymeric membranes, and use of electrorheological fluids in industrial processes.

[1]The first author acknowledges financial support from National Counsel of Technological and Scientific Development (CNPq) and German Academic Exchange Service (DAAD).

9:31AM M26.00008 Laminar Flame Speed of Primary Reference Fuels and Gasoline Surrogates at Elevated Temperatures Measured with the Flat Flame Method[1] , YING-HAO LIAO, WILLIAM ROBERTS, King Abdullah University of Science and Technology — The laminar flame speed is a key target data for validating relevant kinetic mechanisms of the combustion of future fuel formulations since this fundamental parameter contains information for the reactivity, diffusivity, and exothermicity of the fuel mixture. The current work presents the flat flame method, which produces a one-dimensional flat flame free of stretch, to measure laminar flame speeds of the Primary Reference Fuels (PRFs), PRF blends, and gasoline surrogates at elevated temperatures. The flat flame is produced by a McKenna porous plug burner. The laminar flame speed was measured experimentally at atmospheric pressure over a range of equivalence ratios and a range of unburned gas temperatures up to 470 K. To determine the laminar flame speed, a technique with heat extraction through the cooling water, similar to that described by Botha and Spalding (1954), was employed and the adiabatic laminar flame speed was obtained by extrapolation. In addition, the experimental data is compared to simulations using kinetic mechanisms available in the literature. Preliminary results of laminar flame speeds for methane/air and n-heptane/air mixtures at room temperature show good agreement with both of experimental and numerical data available in the literature.

[1]Clean Combustion Research Center

Tuesday, November 26, 2013 8:00AM - 9:57AM –

Session M28 Industrial Applications I Spirit of Pittsburgh Ballroom B/C - Hazel Marie, Youngstown State University

8:00AM M28.00001 Keeping a surface ice/frost free with electro-conducting water-repellent coatings[1] , ARINDAM DAS, SHREYAS KAPATRAL, CONSTANTINE M. MEGARIDIS, Mechanical and Industrial Engineering, University of Illinois at Chicago — Ice/frost formation on aircraft, wind turbines, power grids, marine vessels, telecommunication devices, etc. has propelled scientific research on surfaces that facilitate the removal of the water solid phase or retard its formation. Superhydrophobic, self-cleaning surfaces have been investigated recently (Jung et al., Langmuir 2011) for their passive anti-icing properties. Although superhydrophobic surfaces have been shown to delay the onset of frosting and icing, they cannot prevent it entirely. Hence active deicing/defrosting approaches are required to keep surfaces free of ice/frost. Defrosting experiments have been carried out on glass substrates coated with textured polymeric nanocomposite films of different surface wettability, porosity and roughness. A strong influence of these parameters on condensation, condensation frosting and defrosting was observed. The coatings are electro-conducting, thus allowing skin heating at the interface between ice and the substrate. Sustained ice- and frost-free operation is demonstrated at substrate temperatures well below the freezing point and in humid ambient atmospheres.

[1]Supported by NSF Grant CBET-1066426

8:13AM M28.00002 One-way water permeable valve via water-based superhydrophobic coatings , JOSEPH E. MATES, CONSTANTINE M. MEGARIDIS, Mechanical and Industrial Engineering, University of Illinois at Chicago — Spray-cast superhydrophobic coatings have shown promise in commercial applications for fluid management due to their intrinsic low-cost, large-area capabilities and substrate independence (Schutzius et al. 2011). A technique of applying a light ($<$ 2gsm) water-based superhydrophobic coating on inherently hydrophilic cellulosic substrates to generate a preferred directionality for water absorption and transmission is presented. The mechanism described allows water to pass through a thin treated porous substrate in one direction under negligible pressure, but does not allow water to return from the opposite direction unless much greater pressure is applied. This pressure disparity "window" effectively creates a one-way fluid valve, with envisioned applications ranging from personal hygiene products, to oil-water separation and filtration. Combining SEM imaging with theoretical robustness factors (Tuteja et al. 2008), the penetration pressures are found to be tunable for application-specific designs by choosing a substrate based on limiting factors of fiber diameter and spacing. The process can also be modified with the addition of functionalized (e.g. antibacterial, conductive) nanoparticle fillers suited for the desired application.

8:26AM M28.00003 Mixing of two miscible fluids at high Schmidt number[1] , MARK SIMMONS, FEDERICO ALBERINI, University of Birmingham, CHRISTOPHER PAIN, OMAR MATAR, Imperial College London — The blending of two miscible liquids at high Schmidt number is an increasingly common industrial problem: as processes push towards shorter timescales yet the rheology of the fluids becomes increasingly complex. This leads to phenomena which resemble a quasi two-phase system with zero interfacial tension. In this study, we compare experimental results with computational fluid dynamics simulations using unstructured-mesh adaptivity for the flow of two Newtonian, or two power-law fluids through a complex geometry representative of a Kenics type static mixer used in industry. The geometry induces a stretching and folding of the fluid elements which causes exponential growth of the interface length down the geometry. The interfacial topology obtained from the simulations is compared with experiments carried out using Planar Laser Induced Fluorescence (PLIF), which enables the spatial distribution of each phase, and the interfaces between them, to be determined at the outlet of the geometry.

[1]EPSRC Programme Grant EP/K003976/1

8:39AM M28.00004 Internal Concentration Polarization in Asymmetric Membrane in Forward Osmosis System , GABRIELA GADELHA, HERMES GADELHA, NICK HANKINS, University of Oxford — There has been a re-emerging interest in the study of the osmotic-driving desalination process known as Forward Osmosis (FO), due to its potential for significantly lower energy demand. However, the employed asymmetric semi-permeable membranes are notorious for the formation of unstirred boundary layers. These boundary layers may be dilutive or concentrative, causing an undesired decline on the osmotic flux. To date, although several models have been proposed in the literature to describe various applications in membrane separation processes, the fundamental theoretical basis has remained unchanged. Here, we detail an alternative formulation for the solute concentration profile and the water flux decline in terms of the osmotic Peclet number and the dimensionless solute permeability. Our analysis shows that the osmotic potential efficiency and the resulting water flux are inversely related, preventing any simultaneous optimization of the system, i.e. the larger the water flux is, the less osmotically efficient it becomes. We equally investigated the effect of distinct flat-sheet membrane configurations on the water flux. In this case, when the active layer faces the solution of low concentration (feed solution), under normal operations conditions, the water flux can be 60% lower than its counter configuration, when the active layer faces the solution of high concentration (draw solution). Finally, we contrast the theoretical formulation with experiments using inorganic ions and micelle as draw solutions.

8:52AM M28.00005 Optimizing cross-flow-filtration efficacy using variable wall permeabilities , JAMES HERTERICH, IAN GRIFFITHS, ROBERT FIELD, DOMINIC VELLA, University of Oxford — Water filtration systems typically involve flow along a channel with permeable walls and suction applied across the wall. In this "cross-flow" arrangement, clean water leaves the channel while impurities remain within it. A limiting factor for the operation of cross-flow devices is the build-up of a high concentration of particles near the wall due to the induced flow. Termed concentration polarization (CP), this effect ultimately leads to the blocking of pores within the permeable wall and the deposition of a "cake" on the wall surface. Here we show that, through strategic choices in the spatial variations of the channel-wall permeability, we may reduce the effects of CP by allowing diffusion to smear out any build up of particles that may occur. We demonstrate that, for certain classes of variable permeability, there exist optimal choices that maximize the flux of clean water out of a device.

9:05AM M28.00006 Fluid Mechanics of a High Performance Racing Bicycle Wheel , JEAN-PIERRE MERCAT, BRIEUC CRETOUX, FRANCOIS-XAVIER HUAT, MAVIC, France, BENOIT NORDEY, MAXIME RENAUD, ENSAM, France, FLAVIO NOCA, University of Applied Sciences - hepia - Geneva, Switzerland — In 2012, MAVIC released the most aerodynamic bicycle wheel on the market, the CXR 80. The french company MAVIC has been a world leader for many decades in the manufacturing of bicycle wheels for competitive events such as the Olympic Games and the Tour de France. Since 2010, MAVIC has been in a research partnership with the University of Applied Sciences in Geneva, Switzerland, for the aerodynamic development of bicycle wheels. While most of the development up to date has been performed in a classical wind tunnel, recent work has been conducted in an unusual setting, a hydrodynamic towing tank, in order to achieve low levels of turbulence and facilitate quantitative flow visualization (PIV). After a short introduction on the aerodynamics of bicycle wheels, preliminary fluid mechanics results based on this novel setup will be presented.

9:18AM M28.00007 Numerical simualtions of fouling in crude-oil processing[1] , JUNFENG YANG, OMAR MATAR, Imperial College London — The aims of this study are to achieve fundamental understanding of the transfer processes underlying the development of a fouling layer on the inside of heat exchangers used in crude-oil processing. The numerical models developed are based on the solution of the mass and momentum conservation equations, coupled to the energy transport equation. These are complemented by relations that capture the dependence of the layer density, viscosity, and surface tension on temperature. In order to capture ageing effects, the thermal conductivity of the layer is allowed to depend on temperature, and a functional form is chosen for its viscosity that accounts for dynamic structure-building and destruction. Importantly, a chemical equilibria model is used to model the phase behaviour of the oil, and this is also coupled to the governing equations. The turbulence in the fluid phase is modelled using large eddy simulations. Numerical solutions of the model equations are obtained in a channel geometry using a volume-of-fluid approach. Our results capture the complex fouling dynamics that include phase separation, wall-layer deposition, and removal.

[1]Skolkovo Foundation, UNIHEAT project

9:31AM M28.00008 Parametric Study on the Evolution of Thermal Patterns and Coherent Flow Structures in the Rotated Arc Mixer , OZGE BASKAN, MICHEL SPEETJENS, Eindhoven University of Technology, GUY METCALFE, Commonwealth Scientific and Industrial Research Organisation, HERMAN CLERCX, Eindhoven University of Technology — Advective-diffusive scalar transport in spatially or temporally periodic flow fields has been investigated in numerous studies, which exposed that the global transport relies on the kinematic/geometric parameters governing the advection and the ratio between the advective and diffusive time scales. These studies mainly employ numerical/analytical methods. However, experimental analysis remains outstanding. This research concerns an experimental parametric study on the evolution of the thermal patterns in a representative configuration, the Rotated Arc Mixer (RAM), and its correlation with the coherent flow structures. The RAM is an inline mixer composed of a stationary inner cylinder with systematically oriented apertures and a rotating outer cylinder inducing transverse flow at the apertures. Design of the experimental facility is based on a 2D time-periodic simplification of the 3D spatially-periodic RAM, where the cross-sectional progression is represented by the temporal evolution. The test section is a shallow circular tank with apertures on the circumference and motor-driven belts imitate the rotating outer cylinder of the RAM. Circumferential temperature is kept constant via an enclosing hot-water reservoir. The 2D flow and temperature fields are measured by 2D Particle Image Velocimetry and Infrared Thermography and analyzed.

9:44AM M28.00009 Erosion resistance of pipe bends with bio-inspired internal surfaces[1] , CHENGCHUN ZHANG, Key Laboratory of Engineering Bionics, Jilin University, China, OMAR MATAR, Imperial College London — Guided by the structure of a shell surface, a bio-inspired surface is proposed to enhance the erosion resistance of pipe bends carrying crude-oil and sand in the turbulent flow regime. A comparison of the erosion rate between a smooth bend and the bio-inspired one is carried out using numerical simulations: large eddy simulations are used to simulate turbulence, and these are coupled to a discrete element method for the solid particles. The results indicate that the bio-inspired surface can control effectively the liquid-solid flow near the wall, and decrease the particle-wall force. This, then, leads to a reduction in the erosion rate brought about by the sand transported by the crude-oil in the pipe bend.

[1]The China Scholarship Council is gratefully acknowledged.

Tuesday, November 26, 2013 8:00AM - 10:10AM –

Session M30 Instability: General V - Elastic and Pulsating Flows 408 - Tareneh Sayadi, Ecole Polytechnique

8:00AM M30.00001 Saffman-Taylor Instability for a non-Newtonian fluid[1] , PRABIR DARIPA, Texas A&M University — Motivated by applications, we study classical Saffman-Taylor instability involving displacement of an Oldroyd-B fluid displaced by air in a Hele-Shaw cell. The lubrication approximation is used by neglecting the vertical component of the velocity. We obtain an explicit expression of one of the components of the extra-stress perturbations tensor in terms of the horizontal velocity perturbations. The main result is an explicit formula for the growth constant (in time) of perturbations, given by a ratio in which a term depending on the relaxation and retardation (time) constants appears in the denominator of the ratio. This exact result compares extremely well with known numerical results. It is found that flow is more unstable than the corresponding Newtonian case. This is a joint work with Gelu Pasa.

[1]The research has been made possible by an NPRP Grant # 08-777-1-141 from the Qatar National Research Fund (a member of the Qatar Foundation).

8:13AM M30.00002 Influence of Fluid, Solid, and Geometric Parameters on the Fluid-Structure Interaction Response and Stability of Flexible Lifting Surfaces[1] , EUN JUNG CHAE, DENIZ TOLGA AKCABAY, YIN LU YOUNG, University of Michigan — There is an increasing interest to use innovative passive/active flexible lifting surfaces to take advantage of the fluid-structure interaction (FSI) response to improve performance or harvest energy. However, design and testing of flexible lifting surfaces are quite complicated, particularly for lightweight structures in a dense, viscous fluid. The objectives of this work are to (1) investigate the influence of varying fluid, material, and geometric parameters on the FSI response and stability boundaries, and (2) to develop generic parametric maps to facilitate the design of flexible lifting surfaces In particular, the focus is on the influence of solid-to-fluid density ratio, Reynolds number, relative stiffness ratio, and relative excitation frequency ratio on the FSI response and static/dynamic divergence and flutter stability boundaries. The results show that the governing failure mode transitions from flutter to dynamic divergence to static divergence when the solid-to-fluid added mass ratio decreases. In addition, classic linear potential theory is severely under-conservative in predicting the flutter boundary, and cannot predict the transition to dynamic divergence for cases in the low mass ratio regimes due to the strong nonlinear, viscous FSI response that develops when the fluid forces are comparable or greater than the solid forces.

[1]The Office of Naval Research (Grant no. N00014-11-1-0833); the National Research Foundation of Korea (GCRC-SOP Grant no. 2012-0004783)

8:26AM M30.00003 Stability theory for the synchronized waving of marine grass , RAVI SINGH, SHREYAS MANDRE, Brown University, AMALA MAHADEVAN, Woods Hole Oceanographic Instituion, L. MAHADEVAN, Harvard, MAHESH BANDI, Okinawa Institute of Science and Technology - Japan — Synchronized waving of grass blades in the presence of fluid flow has been observed in cases such as wheat field in wind, marine grass in tidal currents. The synchronous motion can have important environmental and ecological impact via mixing of fluid due to waving. When the hydrodynamic and elastic time scales are well separated, this waving is the manifestation of a shear instability. We extend the Orr-Sommerfeld equation for the stability of a shear flow to include a continuum mean-field approximation for the vegetation, thus capturing the essential ingredients for waving. Our model exhibits an hydrodynamic instability due to different amounts of drag experienced by fluid with in and above the grass. We will also present some numerical results exhibiting existence of a threshold flow speed for waving, which have been observed in case of submerged marine vegetation

8:39AM M30.00004 Travelling waves and fold localization in hovercraft seals[1] , ANDREW WIGGINS, STEVE ZALEK, MARC PERLIN, STEVE CECCIO, University of Michigan — The seal system on hovercraft consists of a series of open-ended fabric cylinders that contact the free surface and, when inflated, form a compliant pressure barrier. Due to a shortening constraint imposed by neighboring seals, bow seals operate in a post-buckled state. We present results from large-scale experiments on these structures. These experiment show the hydroelastic response of seals to be characterized by striking stable and unstable post-buckling behavior. Using detailed 3-d measurements of the deformed seal shape, dominant response regimes are identified. These indicate that mode number decreases with wetted length, and that the form of the buckling packet becomes localized with increased velocity and decreased bending stiffness. Eventually, at a critical pressure, travelling waves emerge. To interpret the wide range of observed behavior, a 2-d nonlinear post-buckling model is developed and compared with the experimental studies. The model shows the importance of seal shortening and the buckling length, which is driven by the balance of hydrodynamic and bending energies. Preliminary scaling laws for the fold amplitude and mode number are presented. The experiments may ultimately provide insight into the bedeviling problem of seal wear.

[1]Sponsored by the Office of Naval Research under grant N00014-10-1-0302, Ms. Kelly B. Cooper, program manager

8:52AM M30.00005 Geometric scaling of purely elastic instability in viscoelastic Taylor-Couette flow , CHRISTOF SCHAEFER, Saarland University, Experimental Physics Department, ALEXANDER MOROZOV, University of Edinburgh, School of Physics and Astronomy, CHRISTIAN WAGNER, Saarland University, Experimental Physics Department — The behavior of viscoelastic Taylor-Couette flow, the flow of, e.g., a polymeric fluid between two concentric, rotating cylinders, has been extensively investigated for many years in experiments as well as in theory. In the most simple case of an outer beaker at rest and a rotating inner cylinder with radii R_2 and R_1, respectively, even at negligible Taylor number $Ta = 2\mathrm{Re}^2(R_2 - R_1)/R_1$, the circular Couette (base) flow gets linearly unstable at a critical Weissenberg number $Wi_c = \lambda\dot{\gamma}$, the product of the characteristic polymer relaxation time λ and the (critical) shear rate $\dot{\gamma}_c$. This non-inertial transition to complex flow patterns is purely elastic by nature and the dimensionless criterion by P. Pakdel and G.H. McKinley (JNNFM 67 (1996)) gives a simple, critical condition for its onset. It pictures the competition between viscous shear and elastic normal stresses as well as the influence of polymer relaxation length and curvature of the streamlines. We present a comparative study of the explicit curvature scaling of the onset of elastic instability in the Taylor-Couette flow, including experimental data as well as linear stability analyses and theoretical examinations.

9:05AM M30.00006 ABSTRACT WITHDRAWN –

9:18AM M30.00007 ABSTRACT WITHDRAWN –

9:31AM M30.00008 Transition to turbulence in pulsating pipe flow , BJORN HOF, Institute of Science and Technology Austria, SASCHA WARNECKE, DUO XU, Max Planck Institute for Dynamics and Self-Organization — We report an experimental study of the transition to turbulence in a pulsating pipe flow the most important example of pulsating flows is the cardiovascular system where the onset of fluctuations and turbulence can be a possible cause for various diseases such as the formation of aneurysms. The present study is limited to a straight rigid pipe, sinusoidal modulation of the flow rate and a Newtonian fluid. The dimensionless parameters (Womersley and Reynolds numbers) were chosen to include the parameter range encountered in larger arteries. We observe that at large frequencies the critical point for the onset of turbulence remains completely unaffected by pulsation for all amplitudes investigated (up to 40%). However for smaller frequencies (Womersley numbers below 10) the critical point considerably increases. Furthermore we investigate how the transition scenario is affected for a fixed frequency and increasing amplitudes (approaching oscillatory flow).

9:44AM M30.00009 Linear stability analysis of pipe Poiseuille flow for an Oldroyd-B fluid , ARMANDOJANNI PETRUCCI OREFICE, GENNARO COPPOLA, LUIGI DE LUCA, Università degli Studi di Napoli "Federico II" — The effects of viscoelasticity on the linear evolution of disturbances on pipe Poiseuille flow are numerically investigated. The viscoelastic fluid is described by the Oldroyd-B model and the work primarily focuses on high Reynolds numbers flows of diluted solutions. The equations governing both flow and elastic variables are written in polar coordinates and are discretized by an accurate Chebyshev pseudospectral code. Both linear modal and non modal stability properties of infinitesimal disturbances are considered. The eigenvalue spectrum of the governing operator and linear transient growth of three dimensional perturbations are determined and the results are compared to analogous classical results for pipe Poiseuille flow. Non modal analysis reveals that elasticity is generally active in reducing the transient growth at high values of streamwise wave number.

9:57AM M30.00010 A Theoretical and Numerical Study of Flexible Flapping Dynamics in a Uniform Flow , RAJEEV JAIMAN, PARDHA GURUGUBELLI, JIE LIU, National University of Singapore — This work presents a numerical and theoretical study of fluid-elastic instability exhibited by a linear elastic plate immersed in a mean flow. Using the Euler-Bernoulli model for the plate and a 2D viscous potential flow model, a generalized closed-form expression of added-mass force has been derived for a flexible plate oscillating in fluid. We present an analytical formulation for predicting critical velocity for the onset of flapping instability. In the second part, a high-order finite element one-field scheme is employed for simulating flapping motion of a thin flexible body in a uniform flow with strong added-mass effects. Through our direct fluid-structure simulations, we study flapping results for a wide range of mass ratios and varying Reynolds numbers while maintaining relatively low bending rigidity. As a function of mass ratio, the flapping dynamics reveals three distinct regimes: fixed-point stability, limit-cycle flapping, and chaotic flapping. The changes associated with regime transition with increasing mass ratio are analyzed by vortex wake patterns, tip displacements, and force coefficients. Dependencies of stability predicated by the theoretical analysis are confirmed by the nonlinear fluid-structure simulations.

Tuesday, November 26, 2013 8:00AM - 9:44AM –

Session M31 Biofluids: Locomotion X - Non-Newtonian Fluids 402 - Thomas Powers, Brown University

8:00AM M31.00001 Enhanced diffusion of tracers in a bath of self-propelled particles[1] , ALEXANDER MOROZOV, University of Edinburgh — Recent experiments have shown that micron-size tracer particles in dilute suspensions of either swimming bacteria or synthetic self-propelled particles perform diffusive motion with the diffusion coefficient significantly larger than its thermal value. Several theories have been proposed to explain the origin and magnitude of the enhanced tracer diffusivity. There is now a general agreement that it is proportional to the so-called "active flux" - the product of the swimmer's number density and their velocity. Here we present detailed theory and simulations of tracers diffusing in bacterial suspensions. Our work confirms the scaling with the active flux, but also unravels new important dependencies on the properties of the velocity field created by the swimmers and their kinematics. Our work is potentially relevant for understanding feeding currents and biomixing created by swimming microorganisms.

[1]Funded by the Engineering and Physical Sciences Research Council (EP/I004262/1).

8:13AM M31.00002 Swimming of wavy sheets in weakly viscoelastic fluids[1] , ALEXANDER MOROZOV, University of Edinburgh — Many natural habitats of biological microswimmers include complex fluids whose mechanical response is strongly non-Newtonian. Recent attempts to understand swimming in such fluids produced a series of seemingly contradictory results. Especially, it is currently not understood whether swimming in dilute polymer solutions would be faster or slower than in Newtonian fluids like water. One of the classical models to study swimming is a 2D infinite periodic waving sheet model introduced by G. I. Taylor. For small-amplitude swimming it was shown previously that viscoelasticity of the suspending fluid reduces the propulsion speed, while simulations of a finite-size version of the same model predicted an increase of the propulsion speed followed by a decrease as the fluid becomes progressively more elastic. Here we present a mechanism for the reduction of the propulsion speed and devise a new swimmer that can go faster in viscoelastic fluids than in their Newtownian counterparts. We perform analytical and exact numerical calculations of large-amplitude swimming of both models to confirm our mechanism.

[1]Funded by the Engineering and Physical Sciences Research Council (EP/I004262/1).

8:26AM M31.00003 Theory for propulsion and transport in an anisotropic fluid , THOMAS POWERS, MADISON KRIEGER, Brown University, SAVERIO SPAGNOLIE, University of Wisconsin, Madison — Swimming microorganisms are typically found in complex fluids, which are full of polymers. When these polymers align, the fluid becomes anisotropic. We seek to understand how anisotropy affects swimming when the stroke is prescribed. We model the anisotropic fluid with a nematic liquid crystal. The swimmer is a two-dimensional sheet deforming via propagating transverse or longitudinal waves. We find that the nature of anchoring conditions for the nematic degrees of freedom plays a critical role in determining the swimming speed. Furthermore, we study the fluid transport induced by the swimmers motion by calculating the flux of fluid in the laboratory frame. Finally, we elucidate the various limits of the nematic theory, such as the six-fold symmetric hexatic case and Ericksen's transversely isotropic fluid.

8:39AM M31.00004 The forward undulatory locomotion of Ceanorhabditis elegans in viscoelastic fluids , AMY SHEN, University of Washington, XIALING ULRICH, Washington University in St. Louis — Caenorhabditis elegans is a soil dwelling roundworm that has served as model organisms for studying a multitude of biological and engineering phenomena. We study the undulatory locomotion of nematode in viscoelastic fluids with zero-shear viscosity varying from 0.03–75 Pa·s and relaxation times ranging from 0–350 s. We observe that the averaged normalized wavelength of swimming worm is essentially the same as that in Newtonian fluids. The undulatory frequency f shows the same reduction rate with respect to zero-shear viscosity in viscoelastic fluids as that found in the Newtonian fluids, meaning that the undulatory frequency is mainly controlled by the fluid viscosity. However, the moving speed V_m of the worm shows more distinct dependence on the elasticity of the fluid and exhibits a 4% drop with each 10-fold increase of the Deborah number De, a dimensionless number characterizing the elasticity of a fluid. To estimate the swimming efficiency coefficient and the ratio $K = C_N/C_L$ of resistive coefficients of the worm in various viscoelastic fluids, we show that whereas it would take the worm around 7 periods to move a body length in a Newtonian fluid, it would take 27 periods to move a body length in a highly viscoelastic fluid.

8:52AM M31.00005 A mechanism for non-Newtonian swimming enhancement , YI MAN, ERIC LAUGA, University of Cambridge — Polymeric solutions and suspensions are prone to display slip due to the presence of thin low-viscosity fluid layers near boundaries. Using theoretical modeling, we investigate the role of such reduction in fluid friction on locomotion of model microorganisms. Addressing two- and three-dimensional situations, we demonstrate how even very thin regions of reduced fluid friction can dramatically enhance locomotion speeds. Our results suggest a mechanism for enhanced swimming in complex fluid environments.

9:05AM M31.00006 Locomotion of microorganisms near a no-slip surface in a viscoelastic fluid , SHAHRZAD YAZDI, Department of Chemical Engineering, The Pennsylvania State University, AREZOO ARDEKANI, Department of Aerospace and Mechanical Engineering, University of Notre Dame, ALI BORHAN, Department of Chemical Engineering, The Pennsylvania State University — Microorganisms are exposed to complex fluids in their natural habitats, especially during biological processes. In many of these processes, microorganisms swim in confined domains such as spermatozoa in mucus of mammalian reproduction tracts or bacteria in extracellular polymeric matrices during biofilm formation. Thus, it is important to understand the kinematics of propulsion in a viscolastic fluid near a no-slip surface. We used a squirmer model with a time-reversible body motion to analytically investigate the swimming kinematics in an Oldroyd-B fluid near a no-slip surface. Our results show that the time-averaged propulsion for a pusher (puller) is towards (away from) the no-slip surface. We present the swimming trajectory as a function of Deborah number, initial distance from the surface, and initial swimming direction.

9:18AM M31.00007 Non-Newtonian rotational swimming: experiments , S. GOMEZ, F.A. GODINEZ, R. ZENIT, Universidad Naciona Autonoma de Mexico, E. LAUGA, University of Cambridge — Recently Pak et al. (PoF, 2012) showed that a device composed of two unequal spheres (snowman) could swim in a viscoelastic fluid under a rotational actuation. By symmetry such device isn't able to move in a Newtonian fluid but because of its geometrical asymmetry is able to generate asymmetric elastic response and generate a purely viscoelastic thrust. We implemented this swimmer experimentally using a magnetic snowman driven by an external rotating magnetic field. We demonstrate that the snowman swims solely as a result of fluid elasticity. We conduct tests in Newtonian and Boger fluids, varying the sphere size ratio and rotation speed. We also conducted measurements in a confined environment, which showed an improved swimming performance.

9:31AM M31.00008 Undulatory Swimming in Fluids with Polymer Networks[1] , DAVID GAGNON, XIAONING SHEN, PAULO ARRATIA, University of Pennsylvania — In this talk, we systematically investigate the motility behavior of the nematode Caenorhabditis elegans in polymeric solutions of varying concentration using tracking and velocimetry methods. As the polymer concentration is increased, the solution undergoes a transition from the semi-dilute to the concentrated regime, where these rod-like polymers entangle, align, and form networks. Remarkably, we find an enhancement in the nematode's swimming speed of approximately 65 percent in concentrated solutions compared to semi-dilute solutions. Using velocimetry methods, we show that the undulatory swimming motion of the nematode induces an anisotropic mechanical response in the fluid. This anisotropy, which arises from the fluid micro-structure, is responsible for the observed increase in swimming speed.

[1]This work was supported by NSF CAREER (CBET) 0954084

Tuesday, November 26, 2013 8:00AM - 10:10AM –

Session M32 Particle-Laden Flows VI: Direct Simulation and Turbulence Modulation 403 - Said Elghobashi, University of California, Irvine

8:00AM M32.00001 Evaluating multiphase turbulence statistics using mesoscale DNS of gravity-driven particle-laden flows , RODNEY FOX, Ecole Centrale Paris, JESSE CAPECELATRO, OLLIVIER DESJARDINS, Cornell University, LABORATOIRE EM2C-UPR COLLABORATION, SIBLEY SCHOOL OF MECHANICAL AND AEROSPACE ENGINEERING COLLABORATION — Flow instabilities encountered in fluid-particle flows subjected to a body force (i.e., gravity) can lead to mesoscale structures that control the underlying fluid turbulence. The wide range of length and time scales associated with such flows pose great challenges in understanding and predicting key features. In our recent work, the exact Reynolds-average (RA) equations for monodisperse collisional particles in a constant-density fluid were derived. The turbulence model solves for the RA particle volume fraction, the phase-average (PA) particle velocity, the PA granular temperature, and the PA particle turbulent kinetic energy. Unlike most previous derivations, a clear distinction is made between the PA granular temperature, which appears in the kinetic theory constitutive relations, and the particle-phase turbulent kinetic energy, which appears in the turbulent transport coefficients. Fully coupled Eulerian-Lagrangian simulations are used to evaluate the unclosed terms that arise in the RA equations. A two-step filter is employed during the interphase exchange process, providing a separation of length scales between the microscale and mesoscale structures. The same filtering process is used to evaluate turbulence statistics and modeling constants appearing in the RA model.

8:13AM M32.00002 A study of turbulence modulation by particle clusters in dilute and moderately-dilute channel flows using mesoscale DNS , JESSE CAPECELATRO, OLIVIER DESJARDINS, Cornell University, NATIONAL RENEWABLE ENERGY LABORATORY COLLABORATION — This work investigates fluid-particle interactions in turbulent channel flows using highly-resolved Euler-Lagrange simulations. In the dilute regime, particle dynamics are mostly controlled by vortical structures in the flow, and wakes past individual particles can modify the underlying fluid turbulence at the particle scale. At moderate concentrations and mass loadings, flow instabilities may lead to mesoscale structures (i.e., clusters) that control underlying fluid turbulence. A Re=13,500 channel flow is studied in both regimes. It is shown in this work that the fluid turbulence departs significantly from the initially fully-developed turbulent flow when subjected to mean particle volume fractions of 1%, where all unsteady features are generated by the cluster dynamics. To study the effect of gravity on clustering dynamics, simulations are conducted with gravity aligned in the mean flow direction, as well as gravity opposing the mean flow direction (i.e., a riser configuration). Velocity fluctuations and energy spectra are computed for each case, along with higher order Lagrangian statistics including collision frequency, radial distribution function, and particle number density.

8:26AM M32.00003 Modeling low-order structure functions for inertial particles in isotropic turbulence , ANDREW BRAGG, LANCE COLLINS, Cornell University — In this talk we will consider three models for the second order structure function of inertial particle pairs in isotropic turbulence, one by Zaichik *et al.* (New. J. Phys. 11:103018, 2009), the second by Pan *et al.* (J. Fluid. Mech. 661:73, 2010) and the third by Gustavsson *et al.* (Phys. Rev. E. 84:045304, 2011). We find that in general they describe the structure functions in qualitatively similar ways, capturing the influence of the nonlocal dynamics on the formation of caustics and non-smooth scaling behavior in the dissipation range. We then compare the predictions with DNS data and find that although they capture the qualitative behavior of the data consistently, they differ with each other quantitatively, with the theory by Pan *et al.* yielding the closest agreement with the DNS. Finally, we show that a new backward in time dispersion theory we have derived makes improvements to the predictions from the Pan *et al.* theory by improving upon a key closure approximation made in its construction.

8:39AM M32.00004 Near-wall, particle-laden turbulent transport , DAVID RICHTER, University of Notre Dame, PETER SULLIVAN, National Center for Atmospheric Research — We use direct numerical simulation coupled with a Lagrangian point-particle formulation to study turbulent planar Couette flow at friction Reynolds numbers ranging between 125 and 900. Modifications to wall-normal scalar and momentum transport are investigated as a function of the size and concentration of the dispersed phase. Furthermore, the dispersed phase effects are examined as the Reynolds number of the flow is increased. In all cases particle phase is observed to weaken the structures responsible for near-wall transport (e.g. hairpins, quasi-streamwise vortices, etc.); an effect which becomes increasingly pronounced as the Reynolds number increases. Physical explanations for this behavior will be presented.

8:52AM M32.00005 Particle-Resolved Direct Numerical Simulation of a Particle-Laden Mixing Layer[1] , MOHAMMAD MEHRABADI, Iowa State University, SUDHEER TENNETI, Now at CD-Adapco, SHANKAR SUBRAMANIAM, Iowa State University — The stability of a homogeneous gas-solid suspension has been investigated in the context of kinetic theory (Koch, Phys. Fluids, 1990) and the averaged two-fluid equations (Glasser et al, PRL, 1998) by considering perturbation of the number density. Koch's analysis points to the dependence of average drag on average volume fraction as the mechanism for the development of instabilities in the number density. However, the physical origins of instabilities in the number density have not been firmly established through microscale simulations at the scale of individual particles. In this study, particle-resolved direct numerical simulation (PR-DNS) is used to ascertain the exact physical origins of the growth of number density instabilities in a particle-laden mixing layer. Self-similarity of the temporally evolving number density profile, and the diffusive/convective nature of the instability is examined to ascertain the role of granular temperature in instability growth. The growth of streamwise and cross-stream structures in the particle field are analyzed to draw analogies with the classical Rayleigh-Taylor and Kelvin-Helmholtz instability mechanisms.

[1]This work is partially supported by NSF CBET 1134500

9:05AM M32.00006 Direct numerical simulation of forward- and backward-in-time relative dispersion of inertial particles in high-Reynolds-number ($R_\lambda \approx 580$) turbulence[1] , PETER IRELAND, ANDREW BRAGG, LANCE COLLINS, Cornell University — Turbulence-induced water droplet coalescence is considered to be an important factor in the onset of precipitation in warm cumulus clouds. Theory (Bragg and Collins 2013) shows that the collision kernel for suspended droplets in turbulence is fundamentally related to their backward-in-time relative dispersion, which has yet to be systematically investigated. Using direct numerical simulations on a 2048^3 lattice with $R_\lambda \approx 580$, we find that inertial particles, like fluid particles, separate more quickly backward than forward in time. However, the degree of asymmetry in the dispersion is significantly greater for inertial particles than for fluid particles. We present new parameterizations for both short and long time relative dispersion and discuss the physical mechanisms leading to the strong asymmetry in the dispersion processes. The results from this work will be used to the improve the theoretical model for particle relative velocities developed by Pan and Padoan (2010), enabling more accurate predictions of collisional droplet growth rates.

[1]This work was supported by NSF grant CBET-0967349. The simulations were performed on NCAR's Yellowstone that is also supported by the NSF.

9:18AM M32.00007 DNS of fully-resolved droplet-laden decaying isotropic turbulence[1] , A. FERRANTE, M. DODD, University of Washington, Seattle — We investigated the effects of finite-size droplets on decaying isotropic turbulence by performing direct numerical simulation (DNS). We performed DNS using our new pressure-correction/volume-of-fluid method that is mass-conservative and second-order accurate. The simulations were performed at $\mathrm{Re}_{\lambda 0} = 75$ on a 1024^3 grid such to resolve each droplet with 32 grid points per diameter. We fully resolve all the relevant scales of turbulence around thousands of freely-moving droplets of Taylor length-scale size as well as the fluid motion inside the droplets. We will discuss the effects of the droplets on the temporal development of turbulence kinetic energy and its dissipation rate. Also, we will present the effects on turbulence of the droplet Weber number and of the density ratio between the droplet and the surrounding fluid.

[1]NSF CAREER #1054591

9:31AM M32.00008 DNS of droplet motion in a turbulent flow[1] , MICHELE ROSSO, S. ELGHOBASHI, Department of Mechanical and Aerospace Engineering, University of California, Irvine — The objective of our research is to study the multi-way interactions between turbulence and vaporizing liquid droplets by performing direct numerical simulations (DNS). The freely-moving droplets are fully resolved in 3D space and time and all the relevant scales of the turbulent motion are simultaneously resolved down to the smallest length- and time-scales. Our DNS solve the unsteady three-dimensional Navier-Stokes and continuity equations throughout the whole computational domain, including the interior of the liquid droplets. The droplet surface motion and deformation are captured accurately by using the Level Set method. The pressure jump condition, density and viscosity discontinuities across the interface as well as surface tension are accounted for. Here, we present only the results of the first stage of our research which considers the effects of turbulence on the shape change of an initially spherical liquid droplet, at density ratio (of liquid to carrier fluid) of 1000, moving in isotropic turbulent flow. We validate our results via comparison with available expe

[1]This research has been supported by NSF-CBET Award 0933085 and NSF PRAC (Petascale Computing Resource Allocation) Award

9:44AM M32.00009 Effect of flow straining on particle accelerations and distribution[1] , ARMANN GYLFASON, Reykjavik University, CHUNG-MIN LEE, California State University Long Beach, LAHCEN BOUHLALI, Reykjavik University, FEDERICO TOSCHI, Eindhoven University of Technology — We explore effects of large scale straining on Lagrangian properties of particles in turbulence. We perform direct numerical simulations of strained turbulence laden with passive and inertial particles of varied inertia, as well as perform particle tracking velocimetry measurement in the same geometry. From both of these studies we analyze particle acceleration statistics to investigate the effect of large scale flow distortion due to straining, resulting in anisotropy that ranges from the large scale down to the inertial range and the dissipative range. A secondary objective is to understand the effects of weak straining on the distribution of particles in the fluid, by examining the evolution of spatial distribution statistics and investigating particle dispersion from simple sources. Particular attention is given to the dependence of these statistics on Reynolds number and rate of strain, in combination with particle inertia.

[1]Icelandic Research Fund Project #080033021

9:57AM M32.00010 Segregation of heavy particles by gravitational force[1] , YONGNAM PARK, CHANGHOON LEE, Yonsei University, Seoul, Korea — The effects of gravitational force on the segregation of heavy particles are investigated in forced isotropic turbulence using direct numerical simulation. The mechanism of preferential concentration of heavy particle has been known to be strongly related with the vortical structures of background turbulence when gravity is not considered. The degree of preferential concentration is maximized when the characteristic time scale of a particle is comparable with the Kolmogorov time scale. In this study, we discover that strong gravity causes a different kind of preferential concentration for high Stokes number particles. Such phenomenon does not seem to be related with the vortical structures. In order to provide a plausible explanation, we investigate the statistics of horizontal divergence at the location of heavy particles. Particles tend to be segregated more as the particles experience longer time of negative divergence, meaning converging motion in the horizontal plane. The ratio of mean duration time of negative divergence to that of positive divergence increases with gravitational force for the high Stokes number particles. More detailed statistics and relevant explanation will be presented in the meeting.

[1]This work was supported by the National Research Foundation of Korea(NRF) grant funded by the Korea government(MSIP) (No.2009-0083510)

Tuesday, November 26, 2013 8:00AM - 10:10AM –

Session M33 Drops XIII: Drop Impact on Dry Surfaces 404 - Irmgard Bischofberger, University of Chicago

8:00AM M33.00001 Fast microdroplet impact: a high-detail investigation using novel experimental methods , CLAAS VISSER, University of Twente, PHILIPP FROMMHOLD, University of Goettingen, SANDER WILDEMAN, CHAO SUN, DETLEF LOHSE, University of Twente, PHYSICS OF FLUIDS GROUP, UNIVERSITY OF TWENTE TEAM, DRITTES PHYSIKALISCHES INSTITUT, UNIVERSITY OF GOETTINGEN TEAM — Optimization of everyday applications (e.g. diesel engines and spray cleaning) requires full control of high-speed microdroplet impact. However, experimental data in this regime is scant. We present a novel method to visualize impact of droplets with a diameter of 50 micometers on hydrophillic and -phobic surfaces, at frame rates beyond 10Mfps. This allowed us to study in high detail the impact dynamics for velocities up to 50 m/s. In addition, the exact droplet shape during spreading was determined by a bottom-view interferometry technique. The study was complemented with numerical simulations, providing a complete and detailed picture of the 3D-flow field during impact. The physics of microdroplet spreading are scale-independent (they are governed by the Weber- and Reynolds numbers alone). This allows to describe and optimize the impact of droplets in a range of industrial applications.

8:13AM M33.00002 Drop splash on a smooth, dry surface , GUILLAUME RIBOUX, JOSE MANUEL GORDILLO, Universidad de Sevilla, ALEXANDER KOROBKIN, University of East Anglia — It is our purpose here to determine the conditions under which a drop of a given liquid with a known radius R impacting against a smooth impermeable surface at a velocity V, will either spread axisymmetrically onto the substrate or will create a splash, giving rise to usually undesired star-shaped patterns. In our experimental setup, drops are generated injecting low viscosity liquids falling under the action of gravity from a stainless steel hypodermic needle. The experimental observations using two high speed cameras operating simultaneously and placed perpendicularly to each other reveal that, initially, the drop deforms axisymmetrically, with $A(T)$ the radius of the wetted area. For high enough values of the drop impact velocity, a thin sheet of liquid starts to be ejected from $A(T)$ at a velocity $V_{jet} > V$ for instants of time such that $T \geq T_c$. If V_{jet} is above a certain threshold, which depends on the solid wetting properties as well as on the material properties of both the liquid and the atmospheric gas, the rim of the lamella dewets the solid to finally break into drops. Using Wagner's theory we demonstrate that $A(T) = \sqrt{3RVT}$ and our results also reveal that $T_c \propto We^{-1/2} = (\rho V^2 R/\sigma)^{-1/2}$ and $V_{jet} \propto We^{1/4}$.

8:26AM M33.00003 Drop splash on a dry, smooth surface: theory , JOSE MANUEL GORDILLO, GUILLAUME RIBOUX, Universidad de Sevilla, ALEXANDER KOROBKIN, University of East Anglia — In this presentation we develop a theoretical model that faithfully predicts, in wide ranges of values of the Ohnesorge and Reynolds numbers, the initial instant at which a high speed sheet is ejected as a result of the impact of a drop onto a dry, smooth substrate. Moreover, the model is able to faithfully reproduce the temporal evolution of the tip of the sheet. We also find that, while the role of the entrapped air bubble can be neglected in the sheet ejection process, the role of air is critical in the dewetting process of the tip of the sheet from the substrate. The splash transition predicted in the Re-Oh and gas to liquid viscosity ratio, agree well with experimental observations.

8:39AM M33.00004 Numerical Simulation of Droplet Impact on Dry Solid Surfaces Using the Moment of Fluid Method , YISEN GUO, YONGSHENG LIAN, University of Louisville, MARK SUSSMAN, Florida State University — The impact of liquid droplets on solid surfaces is a ubiquitous phenomenon in nature and industries. The understanding of the underlying physics involved is critical to many industrial problems such as spray cooling, ink-jet printing, and fuel injection. In this work, we study the droplet impacts on solid surfaces using a Navier-Stokes solver based on the moment of fluid surface representation method. Both dynamic contact angle model and static contact angle model are used. The impacts on both hydrophobic and normal substrates are simulated. The droplet spreading, receding, and rebounding are investigated. Numerical results are compared with experimental results in terms of the droplet base diameters and droplet shapes. Our simulations show that the numerical method can accurately capture the droplet impact phenomena. The simulations also indicate that the dynamic contact models give better match than the static contact angle model.

8:52AM M33.00005 Viscous boundary layer in splashing drops , MICHAEL CHEMAMA, Harvard University, RAVI SINGH, Brown University, MICHAEL BRENNER, Harvard University, SHREYAS MANDRE, Brown University — The discovery that ambient pressure could control the splash of a drop on a solid surface generated renewed efforts to understand the physical mechanisms at work. A recent theoretical analysis [Mandre and Brenner; JFM, 690, 148, (2012)] predicted an initial self-similar evolution governed by the drop's inertia and the viscous drainage of the thin layer of air below. This solution breaks down after surface tension or non-linear inertia terms become important. Viscous effects in the drop were computed and shown to be asymptotically negligible. Here we show that the viscous boundary layer approximation, on which this result relies, can become invalid as there is a crossover between the boundary layer thickness and the typical dynamical length of the self-similar evolution. Whether this happens before or after surface tension sets in can lead to different behaviors.

9:05AM M33.00006 Swirls and splashes: pressure dependence of the airflow created by drop impact[1] , IRMGARD BISCHOFBERGER, KELLY W. MAUSER, JFI and Department of Physics, The University of Chicago, Chicago, IL 60637, BAHNI RAY, TAEHUN LEE, Department of Mechanical Engineering, CCNY, NY 10031, SIDNEY R. NAGEL, JFI and Department of Physics, The University of Chicago, Chicago, IL 60637, JFI AND DEPARTMENT OF PHYSICS, THE UNIVERSITY OF CHICAGO, CHICAGO, IL 60637 COLLABORATION, DEPARTMENT OF MECHANICAL ENGINEERING, CCNY, NY 10031 COLLABORATION — A drop impacting a solid surface with sufficient velocity will splash and emit many small droplets. However, removing the ambient air suppresses splashing completely. The transition between splashing and non-splashing occurs gradually: decreasing the air pressure systematically delays and eventually fully inhibits the occurrence of a splash. The mechanism by which the surrounding gas affects the drop dynamics remains unknown. We use modified Schlieren optics combined with high-speed video imaging to visualize the airflow created by the rapid spreading of the drop after it hits the substrate. We observe the generation of a vortex ring that is initially bound to the outer edge of the spreading liquid and subsequently detaches from the liquid to form a beautiful toroidal vortex sheet that expands and curls up into a roll. We have studied the dynamics of this vortex as a function of gas pressure and find that the sheet gets progressively smaller as the air pressure is decreased. This suggests a weakening of the vortex strength at low pressure.

[1]We acknowledge support from NSF MRSEC and PREM grants.

9:18AM M33.00007 Janus surfaces reveal the hidden face of splashing , ANDRZEJ LATKA, MICHELLE DRISCOLL, SIDNEY NAGEL, James Franck Institute, University of Chicago, Chicago, Illinois 60637, USA — When a drop impacts a dry solid surface, a rapidly moving contact line is created. Subsequently, a thin liquid sheet is ejected from the vicinity of this contact line. The thin sheet then breaks apart to form a splash. Previous work has shown that if the solid surface has a micron-scaled roughness, the thin sheet fails to eject and splashing is suppressed. A striking phenomenon can be observed if the drop impacts a hybrid surface comprised of a rough region, where the initial liquid-solid contact takes place, and a smooth region that is reached by only part of the spreading drop: splashing occurs only where the liquid-solid contact line encounters the smooth surface. Consequently, one observes part of the drop splashing, while the other part spreads on the rough surface undisturbed! The splashing outcome is sensitive to the location of the roughness boundary. Crucially, if the velocity of the contact line as it crosses over into the smooth region is below a threshold velocity, u_{stop}, the drop will not splash even though it has left the rough surface. We describe how this hitherto unidentified characteristic velocity depends on other experimental parameters, such as the liquid viscosity and ambient gas pressure, and discuss the insights it provides into the physical mechanisms underlying splashing.

9:31AM M33.00008 Multiscale liquid drop impact on wettable and textured surfaces , SAMANEH FAROKHIRAD, Department of Mechanical Engineering, City College of City University of New York, RUI ZHANG, JOEL KOPLIK, Benjamin Levich Institute and Department of Physics, City College of City University of New York, TAEHUN LEE, Department of Mechanical Engineering, City College of City University of New York — We present the impact of microscopic liquid droplets on solid surfaces which are flat, or pillared, with either homogeneous interactions or cross-shaped patterns of wettability using numerical simulations. The focus is on relatively low impact velocities leading to spreading or bouncing drops, rather than splashing. Lattice Boltzmann and Molecular dynamics methods are used for nanometer-sized and continuum droplets, respectively, and the results of the two methods are compared in terms of scaled variables. In most situations we find similar droplet behavior at both length scales. The agreements between the methods are reasonable at low impact velocities on wettable surfaces while some discrepancies are observed for strongly hydrophobic surfaces and for higher velocities.

9:44AM M33.00009 Disintegration of a Round Liquid Jet due to Impact on a Superhydrophobic Surface , MAZIYAR JALAAL, BORIS STOEBER, Department of Mechanical Engineering, The University of British Columbia, Vancouver, BC, Canada — Liquid jet breakup has several applications such as Inkjet printers, diesel fuel injectors, and paint sprays. Very recently liquid jets have been shown to be useful for small volume transportation (*Clestini et al. Soft Matter, 2010*), where a micro-scale liquid jet on superhydrophobic surface was investigated. Although the instability of the liquid jet for some circumstances was shown, the disintegration of the liquid jet was not discussed. In the present study, we aim to analyze the breakup of a micro liquid jet due to inclined impact to a superhydrophobic surface. A range of Weber and Reynolds numbers have been explored experimentally. Water-glycerin solution as the working fluid. Generally, it is shown that the liquid jet forms a disc-like film over the surface and further rebounds ("bouncing jet"). A simple energy balance method is provided to estimate the diameter of the disc-like film. It is shown, for the case of low viscosity (large Re), this parameter is logarithmically proportional to the normal Weber number. Additionally, linear stability analysis for viscous jets provides a good estimate of droplet size. From an application point of view, using superhydrophobic surfaces 1) enables rebound of the liquid jet 2) advances the breakup point (shorten the breakup length).

9:57AM M33.00010 Drop impacts on electrospun nanofiber membranes[1] , RAKESH P. SAHU, SUMAN SINHA-RAY, ALEXANDER YARIN, University of Illinois at Chicago, BEHNAM POURDEYHIMI, North Carolina State University — This work reports a study of drop impacts of polar and non-polar liquids onto electrospun nanofiber membranes (of 8–10 mm thickness and pore sizes of 3–6 nm) with an increasing degree of hydrophobicity. The nanofibers used were electrospun from polyacrylonitrile (PAN), nylon 6/6, polycaprolactone (PCL) and Teflon. It was found that for any liquid/fiber pair there exists a threshold impact velocity (1.5 to 3 m/s) above which water penetrates membranes irrespective of their wettability. The low surface tension liquid left the rear side of sufficiently thin membranes as a millipede-like system of tiny jets protruding through a number of pores. For such a high surface tension liquid as water, jets immediately merged into a single bigger jet, which formed secondary drops due to capillary instability. An especially non-trivial result is that superhydrophobicity of the porous nano-textured Teflon skeleton with the interconnected pores is incapable of preventing water penetration due to drop impact, even at relatively low impact velocities close to 3.46 m/s. A theoretical estimate of the critical membrane thickness sufficient for complete viscous dissipation of the kinetic energy of penetrating liquid corroborates with the experimental data.

[1]The current work is supported by the Nonwovens Cooperative Research Center (NCRC).

Tuesday, November 26, 2013 8:00AM - 10:10AM –

Session M34 Drops XIV: Shape Dynamics and Confinement 405 - Petia Vlahovska, Brown University

8:00AM M34.00001 Nonlinear Resonance of Mechanically Excited Sessile Drops , CHUN-TI CHANG, SUSAN DANIEL, PAUL STEEN, Cornell University — The spectrum of frequencies and mode shapes for an inviscid drop on a planar substrate have recently been documented. For vertical excitation, zonal modes respond to the driving frequency harmonically and non-zonal modes subharmonically, consistent with the prior literature. In this study, we report observations from the regime of nonlinear response. Here, zonals can respond non-harmonically, both sub- and super-harmonic responses are reported. The principal challenge to generating and observing superharmonic resonances of higher zonal modes is a mode-mixing behavior. However, using a simple visual simulation based on the ray-tracing technique, the individual contributions to the mixed resonance behavior can be extracted. In summary, results from experiment and theory show that the zonal modes, which respond harmonically and can mix with non-zonal modes without interfering with one another in the linear regime, tend to respond sub- or superharmonically and compete with non-zonal modes in the nonlinear regime.

8:13AM M34.00002 Experiments on the harmonic response of coupled droplets to pressure forcing , CHRISTOPHER TILGER, JOSEPH OLLES, AMIR HIRSA, Rensselaer Polytechnic Institute — The dynamic response of a coupled droplet system to sinusoidal forcing pressure was studied. The system consists of two droplets, each with pinned contact lines, coupled through a conduit. Applications of the coupled droplet system include fast focusing liquid lenses and liquid adhesion devices. The millimeter-scale pinned-contact lines associated with the coupled droplets minimize viscous dissipation and allow capillarity to maintain spherical caps. Phase lag, averaged over the period, between the motion of the droplets and the pressure signal was measured using a high speed imaging system. The instantaneous phase behavior of the system was also determined, describing the droplet configuration with respect to the input forcing. Finally, oscillation lag times associated with starting and stopping transients were determined along with damping coefficients for coupled droplets of different volumes.

8:26AM M34.00003 Shape deformation dynamics of acoustically pulsed functional pendant droplet undergoing burning , SAPTARSHI BASU, ANKUR MIGLANI, Indian Institute of Science, RANGANATHAN KUMAR, University of Central Florida — Understanding surface dynamics at the droplet scale is a problem of fundamental significance and general utility. We show that the preferential entrapment driven homogenous boiling in burning functional droplet can induce severe bulk shape oscillations in the droplet. Internal pressure upsurge resulting from ebullition activity force ejects bubble from the droplet domain causing surface undulations and oscillations in bulk, thus driving the droplet into a nonperiodic swell-shrink cycle. The extent of droplet deformation depends on the frequency and intensity of these bubble expulsion events. Besides, the bubble ejections result in localized droplet fragmentation with the subsequent formation of pinched-off satellite droplets that aids secondary atomization. In a unique regime of single major bubble residing within the droplet the pre-ejection transient time is characterized by an interfacial DL instability, where volumetric bubble-shape oscillations drive the droplet oscillations. However, in the presence of longitudinal acoustic forcing at 100 Hz this instability appears to be suppressed since external pulsing modulates the droplet flame into an oscillatory heat source, resulting in delayed bubble incipience and reduced bubble growth rates.

8:39AM M34.00004 Manipulating the breakup dynamics of a droplet by lading nanoparticles in the liquid phase , SAPTARSHI BASU, DEEPU P, SHUBHAM CHOWDHURI, Indian Institute of Science — The deformation and breakup characteristics of a 5 μl sessile droplet excited via support motion are studied by employing high speed imaging. The support was actuated in a sinusoidal fashion using electromagnetic means at different frequencies and amplitudes. It is observed that under resonant conditions, the droplet shows vigorous oscillations and eventually disintegrates. Introducing nanoparticles into the liquid phase is seen to suppress the breakup of the droplet. By studying the oscillation behavior of glycerol-water mixtures at different concentrations, the effect of the presence of nanoparticles is established to be tantamount to increasing the fluid viscosity. Frequency spectra of the response (quantified in terms of the droplet height) of the different droplets revealed that higher modal excitation of the droplet is suppressed with increasing viscosity. This non-linear interaction among the higher harmonics explains the increasing trend of phase lag between the driving force and the response as the viscosity increases. A theoretical model based on proper orthogonal decomposition was developed which corroborates all the experimental trends.

8:52AM M34.00005 Water Drop Shedding under Icing Conditions from Surfaces with Different Wettabilities , DEEPAK KUMAR MANDAL, York University, ANTONIO CRISCIONE, Technical University of Darmstadt, ALIDAD AMIRFAZLI, York University, TECHNICAL UNIVERSITY OF DARMSTADT TEAM, YORK UNIVERSITY TEAM — A sessile water drop on a substrate exposed to airflow will shed if the adhesion force is overcome by external forces on the drop. There are a number of theoretical and experimental studies examining the shedding of drops as described in the normal laboratory conditions. Drop shedding under icing conditions is not well understood; this is not only important from fundamental perspective, but also for technological applications such as icing of aerodynamic surfaces (e.g. wings of an aircraft). An icing wind tunnel was designed where both airflow and surface temperature (system temperature) can be lowered up to -10 °C. Drop shedding on surfaces with different wetting characteristics (from hydrophilic to superhydrophobic) were investigated to determine critical air velocity at which the drop starts to shed. Water drops of different volumes (5 - 100 μl) were used to analyze the influence of the drop volume on the critical air velocity for shedding on cold surfaces. Results show that the system temperature and wetting properties have a major influence on drop shedding under icing conditions. The critical velocity for drop shedding decreases as the volume of the droplet increases for a particular surface. The influence of different surface and ambient temperature on the shedding of the droplet will be presented.

9:05AM M34.00006 Oil droplet behavior at a pore entrance in the presence of crossflow: Implications for microfiltration of oil-water dispersions , TOHID DARVISHZADEH, VOLODYMYR TARABARA, Michigan State University, NIKOLAI PRIEZJEV, Wright State University — The behavior of an oil droplet pinned at the entrance of a micropore and subject to clossflow-induced shear is investigated numerically by solving the Navier-Stokes equation. We found that in the absence of crossflow, the critical transmembrane pressure required to force the droplet into the pore is in excellent agreement with a theoretical prediction based on the Young-Laplace equation. With increasing shear rate, the critical pressure of permeation increases, and at sufficiently high shear rates the oil droplet breaks up into two segments. The results of numerical simulations indicate that droplet breakup at the pore entrance is facilitated at lower values of the surface tension coefficient, higher oil-to-water viscosity ratio and larger droplet size but is insensitive to the value of the contact angle. Using simple force and torque balance arguments, an estimate for the increase in critical pressure due to crossflow and the breakup capillary number is obtained and validated for different viscosity ratios, surface tension coefficients, contact angles, and drop-to-pore size ratios.

9:18AM M34.00007 Oval track droplets racing to a circle: a generic behavior for confined droplets relaxation and a geometrical model[1] , PIERRE-THOMAS BRUN, MATHIAS NAGEL, FRANCOIS GALLAIRE, EPFL / LFMI — Working in a Hele-Shaw cell, the ideal case of the relaxation of a flattened cylindrical droplet of apparent elliptical cross section is considered. Even though the typical Reynolds number imposed by the problem size is extremely low we found out that the investigated pancake droplet relaxes in a remarkable non-monotonous way. After a transient regime, where the droplet adopts a "peanut" shape, it relaxes among a novel family of ovals that to our knowledge has never been reported. These shapes, further referred to as κ_2 ovals, are recovered from geometrical constrains that arise from a linear stability analysis. Far from being limited to initially elliptical relaxing droplets the κ_2 ovals appear to be generic and are found in the relaxation of any initially symmetrical shape. This point is well understood thanks to the previously evoked linear stability analysis. A practical example of such relaxations is provided when considering the coalescence of two identical droplets. Experimental movies of FC40 oil droplets in a water continuous phase are provided. Coalescence is achieved in a PDMS micro-channel and the obtained images are used for comparison with our theoretical work.

[1]This work was supported by the ERC SIMCOMICS.

9:31AM M34.00008 Simulations of drop transport through obstacle arrays , RUI ZHANG, JOEL KOPLIK, City College of CUNY, KOPLIK TEAM — Motivated by possible applications to the separation of deformable cells or drops, we use molecular dynamics simulations to investigate the transport properties of liquid drops in a periodic lattice of cylindrical posts at capillary numbers O(1) and Reynolds numbers O(10), and compare the results to previous studies for rigid or weakly deformable particles. A drop impacting a single obstacle is observed to deform, deflect, split and/or recombine, depending on the incident velocity and impact parameter, as well as the degree of mixing with the carrier fluid. We characterize the collision outcome and trajectory deflection as a function of these parameters. The calculations are extended to a periodic array of cylindrical obstacles, where we focus on the survival probability, drop size distribution and trajectory deflection. The results are compared to the directional locking effect observed for rigid particles in obstacle arrays.

9:44AM M34.00009 New Large Length Scale Capillary Fluidics Investigations Using a Drop Tower[1] , MARK WEISLOGEL, ANDREW WOLLMAN, BRENTLEY WILES, Portland State University — Drop Towers provide brief terrestrial access to microgravity environments. When exploited for capillary fluidics research, the drop tower allows for unique control over an experiment's initial conditions which can enable, enhance, or otherwise improve methods to study capillary flows and phenomena at significantly larger length scales than can be achieved on the ground. In this work a new, highly accessible, 2.1 s tower is introduced for such research. Enabled in part by simple macro-fabrication methods, a variety of new demonstrative experiments are presented for purely capillarity-driven flows leading to droplet ejections, bubble ingestions, sinking flows, particle injections, and multiphase flows. Due to the repeatability of the passive flows, each experiment may be used in turn as a means to study other phenomena and forward-looking research themes are suggested that include large length scale passive phase separations, heat and mass transfer, droplet dynamics, combustion, and more.

[1]NASA NNX09AP66A Glenn Research Center, NASA NNX10AK68H Oregon Space Grant Consortium

9:57AM M34.00010 Zero-gravity mean free surface curvature of a confined liquid in a radially-vaned container[1] , YONGKANG CHEN, Portland State University, MICHAEL CALLAHAN, Jacobs Technology, MARK WEISLOGEL, Portland State University — A variety of increasingly intricate container geometries are under consideration for the passive manipulation of liquids aboard spacecraft where the impact of gravity may be neglected. In this study we examine the mean curvature of a liquid volume confined within a radial array of disconnected vanes of infinite extent. This particular geometry possesses a number of desirable characteristics relevant to waste water treatment aboard spacecraft for life support. It is observed that under certain conditions the slender shape of the free surface approaches an asymptote, which can be predicted analytically using new hybrid boundary conditions proposed herein. This contribution represents possibly the final extension of what has been referred to as the method of de Lazzer et al. (1996). The method enables the integration of the Young-Laplace equation over a domain bounded by the wetted portion of the solid boundaries, symmetry planes, and circular arcs representing free surfaces at the center plane of the liquid body. Asymptotic solutions at several limits are obtained and the analysis is confirmed with numerical computations.

[1]This research is supported in part by NASA Johnson Space Center

Tuesday, November 26, 2013 8:00AM - 9:57AM –

Session M35 Turbulence: Shear Layers II - Experiments 406 - Joseph Katz, Johns Hopkins University

8:00AM M35.00001 Vorticity Based Intermittency in the Single Stream Shear Layer (SSSL) , JOHN FOSS, KYLE BADE, Michigan State University, RICHARD PREVOST, DOUGLAS NEAL, LaVision Inc. — The uniquely large scale and high Reynolds number, Re(θ)=65120, single stream shear layer (SSSL), extensively studied by Morris and Foss (2003), has been investigated using overlapped PIV images. The PIV data focus on the low-speed side: 0<u/U<0.8, of the SSSL. The PIV data complement the hot-wire data which were previously used to extensively examine the high-speed side of the SSSL. (The PIV results overcome significant uncertainty sources of hot-wire anemometry as the time-mean streamwise velocity approaches zero on the low-speed side of the SSSL.) Conventional stochastic values have been obtained using the PIV results. The substantial value of this study is to exploit the unique irrotational state of the entrainment stream, which allows the in-principle designation for intermittency as: I=1 if the point-wise vorticity is non-zero, and I=0 where vorticity equals zero with high spatial resolution. These measures can be used to identify the viscous super-layer (VSL) as the border between the two domains.

Morris, S.C. and Foss, J.F. (November 2003) "Turbulent boundary layer to single-stream shear layer: the transition region," Jour. Fluid Mechanics, 494, pp. 187-221.

8:13AM M35.00002 Coherent structures and momentum transport at various scales above an array of multiscale structures[1] , KUNLUN BAI, JOSEPH KATZ, CHARLES MENEVEAU, Johns Hopkins University — Detailed PIV measurements are carried out to study the turbulence and coherent structures at various scales above a canopy composed of multiscale fractal tree-like objects. The fractal tree has five generations, each consisting of three branches. To study the turbulent structures and momentum transport at large scales, quadrant analysis of fluctuation velocity is carried out. It shows that close to the canopy, sweeping events have larger contribution to the Reynolds shear stress than ejections. Away from the canopy, on the other hand, sweeping contributes less to shear stress than ejection. When the ejection is at present, the flow is disturbed greatly, and the inclined angle of vortices packets can be more than 30 degree. Close to the canopy, the correlation spectrum , i.e. $-E_{uw}/(E_{uu}E_{ww})^{0.5}$ (where E_{uw} is the co-spectrum and E_{uu} and E_{ww} are spectra of streamwise and vertical velocities, respectively), first decreases and then lifts up as wavenumber increases or scale decreases. It indicates the presence of small-scale coherent structures close to the canopy that contribute, at small scales, to momentum transport. A physical space filtering technique is applied to the velocity field to study such structures and the associated large-scale flow patterns.

[1]Acknowledgement: This research is supported by NSF-AGS-1047550 and the Sardella Chair at Johns Hopkins University.

8:26AM M35.00003 Particle Image Velocimetry of a Supersonic Flow over a Finite-Width Rectangular Cavity , STEVEN BERESH, JUSTIN WAGNER, JOHN HENFLING, RUSSELL SPILLERS, BRIAN PRUETT, Sandia National Laboratories — Stereoscopic particle image velocimetry measurements have been acquired in the streamwise plane for supersonic flow over a rectangular cavity of variable width, peering over the side wall lip to view the depths of the cavity. The complex camera angles were treated using two-axis scheimpflug focusing and perspective correction inherent in the camera calibration algorithm. The data reveal the turbulent shear layer over the cavity and the recirculation region within it. The mean position of the recirculation region was found to be a function of the length-to-width ratio of the cavity, as was the turbulence intensity within both the shear layer and the recirculation region. Compressibility effects were observed in which turbulence levels dropped and the shear layer thickness decreased as the Mach number was raised from 1.5 to 2.0 and 2.5. Supplemental measurements in the crossplane and the planform view suggest that zones of high turbulence were affixed to each side wall centered on the cavity lip, with a strip of turbulence stretched out across the cavity shear layer whose intensity was a function of the length-to-width ratio.

8:39AM M35.00004 Nozzle Turbulent Boundary Layer Influence on Sound in a Mach 0.9 Jet , RYAN FONTAINE, Department of Mechanical Science & Engineering, University of Illinois, GREGORY ELLIOTT, JOANNA AUSTIN, Department of Aerospace Engineering, University of Illinois, JONATHAN FREUND, Departments of Mechanical Science & Engineering and Aerospace Engineering, University of Illinois — One of the principal challenges in the prediction and design of low-noise nozzles is the thinness of the near-nozzle mixing layers at the high-Reynolds-numbers of engineering conditions. The specific challenge will depend in part on the upstream conditions, but typically we expect that the jet leaving the nozzle will have $Re_D \sim 5 \times 10^6$ at application scales. Including this in design approaches presents a significant challenge. It is well known that small-scale experiments will have relatively thicker near-nozzle shear layers, which can hamper their applicability to high-Reynolds-number design. Though they can nominally be run at sufficiently high Reynolds numbers, faithfully representing these regions within a large-eddy simulation is likewise a challenge because the locally largest scales are so small. A family of nozzles designed to change the exit thickness of the turbulent boundary layer with otherwise identical flow conditions is studied experimentally to quantify the sensitivity of the far-field sound to nozzle shear layer conditions, which are quantified with very-near-nozzle PIV measurements. The influence is pronounced, though less significant than the well-known sensitivity of far-field sound to laminar versus turbulent boundary near-nozzle shear layers.

8:52AM M35.00005 An experimental investigation of the shear-layer and acoustic sources produced by a leading edge slat[1] , STEPHEN WILKINS, PATRICK RICHARD, JOSEPH HALL, University of New Brunswick, TURBULENCE AND FLOW NOISE LABORATORY TEAM — Leading edge slats are a common addition to airfoils as part of a high lift configuration employed during take-off and landing; the unsteady flow caused by these slats is a major contributor to the overal airframe noise. As the next generation of aircraft seeks to reduce these noise concerns, a better understanding of the sources of aeroacoustic noise generation is sought. Particle Image Velocimetry (PIV) and simultaneous multipoint measurements of the unsteady surface pressure are used herein to investigate the unsteady flow around a leading edge slat coupled with an airfoil for several different configurations and a range of Reynolds numbers ($Re = 156,000$ to $Re = 1.2$ million based on the wing chord). Shear-layer development off the slat cusp and the related unsteady vortex structures are examined in detail to better establish and understand the mechanisms responsible for the generation of aeroacoustic slat noise.

[1]The authors are grateful for the support provided by GARDN.

9:05AM M35.00006 Vortex Identification above the free-end of finite-height prisms and cylinders , RAJAT CHAKRAVARTY, NOORALLAH ROSTAMY, DONALD BERGSTROM, DAVID SUMNER, University of Saskatchewan — The local flow field above the free end of a surface-mounted finite-height square prism of aspect ratio = AR = 5 was experimentally studied using 2D PIV measurements on the vertical symmetry plane. The velocity field data were post-processed using Proper Orthogonal Decomposition (POD) and swirling strength. POD was performed using 100 instantaneous snapshots to generate a reconstruction of the vorticity field. It was observed that flow separation from the leading edge of the free-end surface led to the formation of a separated shear layer and a mean recirculation zone below it. POD and swirling strength comparisons successfully isolated small-scale vortices at the shear layer interface not resolved in previous studies. These vortices became progressively larger and weaker, finally smearing out into the wake. Swirling strength, though inherently a 3D concept, was able to isolate the corresponding vortex structures even in regions where the 2D local velocity gradient tensor yielded complex eigenvalues. It was concluded that swirling strength can also be used in 2D flow fields on planes of symmetry to isolate vortices whose axes have no component in the mean flow direction.

9:18AM M35.00007 Transition to turbulence in stratified shear flow: experiments in an inclined square duct[1] , COLIN MEYER, PAUL LINDEN, University of Cambridge — We describe laboratory experiments of countercurrent stratified shear flow in an inclined square duct. To achieve this, a long water tank was partitioned into regions of higher and lower density saltwater that are connected by an inclined square duct. The flow regime was characterized to be turbulent, intermittent, Holmboe or laminar as a function of the duct inclination, θ, and the density difference, $\Delta\rho$, between the two reservoirs. The density difference and duct angle were systematically varied and a phase plane of flow regime was developed. The transition between the interrmittent regime and turbulence was experimentally determined to occur at $\theta\Delta\rho \simeq 20$ [degrees kg m^{-3}]. This critical combination of parameters fits into the buoyancy-compensated Reynolds number scaling proposed by Brethouwer et al. (*J. Fluid Mech.*, 2007). The turbulent interfacial thickness was found to be a function of the inclination angle, which can be predicted using the buoyancy lengthscale from Waite and Bartello (*J. Fluid Mech.*, 2004) and others. Furthermore, we measured the density profiles at multiple points along the duct, and using these profiles, we modeled the entrainment at the interface.

[1]Support provided by the Winston Churchill Foundation of the United States

9:31AM M35.00008 3D Evolution of Turbulent Flow Structures in Taylor-Couette , SEDAT TOKGOZ, GERRIT E. ELSINGA, JERRY WESTERWEEL, Lab. for Aero & Hydrodynamics, 3ME Faculty, Delft University of Technology, Delft, The Netherlands — In this study, we use high-speed tomographic PIV to investigate the evolution of turbulent flow structures in Taylor-Couette flow. High-speed tomographic PIV enables fully volumetric time-resolved measurements and is well-suited for this purpose. Presently, the turbulent flow is created by exact counter-rotation of the cylinders ($Re_i = -Re_o$, $R_\Omega = 0.0$), where the wall velocities are the same with opposite sign. Under these circumstances, the mean flow is zero in the bulk. Results indicate that the structures still advance in one direction despite the zero mean. However, the observation time for the flow structures is still at least one order magnitude longer than in the boundary layer flows, which were considered before. Results also revealed the presence of azimuthal velocity streaks. It is found that the intense vortical structures are mostly located in the shear layer between these streaks. Different events regarding the evolution of the vortical structures, such as stretching and break-up of vortices, are observed.

9:44AM M35.00009 The behavior of the wake behind a heated circular cylinder , MORTEZA KHASHEHCHI, KAMEL HOOMAN, The University of Queensland, THE QUEENSLAND GEOTHERMAL ENERGY CENTRE OF EXCELLENCE (QGECE) TEAM — The thermal effects on the characteristics of the wake behind a circular cylinder operating in the mixed convection regime are considered at relatively high Reynolds number using Particle Image Velocimetry. The experiments were conducted in a horizontal wind tunnel with the heated cylinder placed horizontally. With such assumptions, the direction of the thermally induced buoyancy force acting on the fluid surrounding the heated cylinder would be perpendicular to the flow direction. Experiments were conducted for three Reynolds numbers 1000, 2000 and 4000, where each of them were run at three different temperatures 25, 50 and 75°C. By adjusting different temperatures in different Reynolds numbers, the corresponding Richardson number ($Ri_D = Gr/Re^2$) was varied between 0.0 (unheated) and 10, resulting in a change in the heat transfer process from forced convection to mixed convection. With increasing temperature of the heated cylinder, significant modifications of the wake flow pattern and wake vortex shedding process were clearly revealed. In low Richardson number, the size of the wake and the vortex shedding process in the wake was found to be quite similar to that of an unheated cylinder. As the Richardson number increased, the wake vortex shedding process was found to be altered and the relative position of the first detached vortices respect to the second one is changed. It was also found that the shedding frequency of the wake vortex structures and the wake closure length decreased with increasing Richardson number.

Tuesday, November 26, 2013 8:00AM - 9:31AM –

Session M36 Geophysical: Oceanographic VII 407 - Arezoo Ardekani, University of Notre Dame

8:00AM M36.00001 Experiments in Stably Stratified Wakes I: Measurement and Characterization of Mean and Fluctuating Quantities[1] , XINJIANG XIANG, TRYSTAN MADISON, PRABU SELLAPPAN, GEOFFREY SPEDDING, University of Southern California — In a stable background density gradient, initially turbulent motions evolve into a state that is dominated by low Froude number dynamics and that can also contain persistent pattern information. Nevertheless, little quantitative information is available in the initial flow evolution when the turbulence first adjusts to the background. Here we report on experiments in a refractive index matched facility for $0.6 \leq Fr \leq 8$ and $2500 \leq Re \leq 10000$, where flow quantities behind a towed grid are examined, and appropriate measures for this early wake regime are considered.

[1]Support from ONR N00014-11-1-0553 gratefully acknowledged.

8:13AM M36.00002 Experiments in stably stratified wakes II: The early wake behind a sphere[1] , TRYSTAN MADISON, XINJIANG XIANG, PRABU SELLAPPAN, GEOFFREY SPEDDING, University of Southern California — The wake of a towed sphere has been used as a canonical case for investigating turbulence in a stratified environment, and certain late wake features compare well with numerical experiments that have no sphere. As a result empirically-established evolution laws that do not depend on initial or boundary conditions are thought to be quite general. It is just becoming possible, experimentally and numerically, to access the early stages of flow development around the sphere itself, when a much more specific and rigorous comparison of similar quantities can be made. Here the first quantitative early wake data behind a towed sphere in a laboratory experiment for $Fr = \{2, 8\}$ and $Re = \{2500, 10000\}$ are presented.

[1]Support from ONR N00014-11-1-0553 gratefully acknowledged

8:26AM M36.00003 Retention and entrainment effects: experiments and theory for porous spheres settling in sharply stratified fluids[1] , SHILPA KHATRI, ROBERTO CAMASSA, CLAUDIA FALCON, RICHARD MCLAUGHLIN, JENNIFER PRAIRIE, BRIAN WHITE, SUNGDUK YU, Department of Mathematics and Department of Marine Sciences, University of North Carolina at Chapel Hill, UNC JOINT FLUIDS LAB TEAM — Marine snow, porous aggregates composed of phytoplankton, fecal pellets, sediment, detritus and other material found in the ocean, are fundamental to the carbon flux from the surface ocean to the deep ocean. Oceanographers observe that marine snow often accumulate in layers whose location are correlated with sharp density gradients in the water column. Understanding the formation and depletion of these marine layers is important to being able to accurately model the marine carbon cycle. A first step in an ongoing investigation is to study the settling of a single porous particle through ambient density gradients. We have conducted experiments to study the settling behavior of single porous spheres in sharp and linear density gradients. Experimental data are first compared to a model based on diffusive processes. Comparisons show that the model predicts accelerations of the particle but not the retention times accurately. Entrainment of less dense fluid from above is then included in the modeling, which allows retention times to be accurately captured. Entrainment shell thickness as a function of parameters will be discussed.

[1]We acknowledge funding received from the following NSF grants: RTG DMS-0943851, RAPID CBET-1045653, CMG ARC-1025523, and DMS-1009750.

8:39AM M36.00004 Rapid distortion theory for mixing efficiency of a flow stratified by one or two scalars[1] , CHRIS REHMANN, JENNIFER JEFFERSON, Iowa State University — The mixing efficiency of unsheared homogeneous turbulence in flows stratified by one or two active scalars was calculated with rapid distortion theory (RDT). For one scalar the mixing efficiency ηdepends on the Schmidt number and the Grashof number. For two scalars the efficiency also depends on the density ratio R_ρ, which compares the density differences caused by temperature and salt. In the one scalar case when *Gr* is large, η decreases as *Sc* increases. The mixing efficiency increases with *Gr* up to a maximum value, as in numerical simulations and experiments. The maximum of approximately 30% for low *Sc* is consistent with simulations, while the maximum of 6% for heated water is consistent with laboratory measurements. However, RDT underpredicts the maximum for saltwater and the value of *Gr* at which the efficiency becomes constant. For two active scalars, η decreases as R_ρ decreases, as in experiments. Results from simulations with low *Sc* likely overestimate the efficiency of turbulence in strongly stratified flows in lakes and oceans.

[1]We acknowledge support from the U.S. National Science Foundation.

8:52AM M36.00005 Reorientation of elongated particles at density interfaces , AMIN DOOSTMOHAMMADI, AREZOO ARDEKANI, University of Notre Dame — The settling rates of particles and organisms in oceanic environments can be considerably affected by approaching density interfaces. The presence of density gradients have been correlated to important environmental phenomena such as accumulation of marine snow particles and intense biological activities. Although many of these settling particles and organisms are of elongated shapes, the current knowledge of settling through density interfaces centers around spherical particles. Here, we uncover the role of the density gradient in changing the orientation of elongated particles. By using direct numerical simulations, we demonstrate that unlike the homogeneous fluid, the presence of density gradients tend to turn the elongated particle so that its broadside is parallel to the direction of the gravity. We provide a phenomenological description of the underlying physics by characterizing deflection of isopycnals and generation of buoyancy induced vortices.

9:05AM M36.00006 Stratified mixing by microorganisms , GREGORY WAGNER, WILLIAM YOUNG, University of California, San Diego, ERIC LAUGA, DAMTP, University of Cambridge — Vertical mixing is of fundamental significance to the general circulation, climate, and life in the ocean. In this work we consider whether organisms swimming at low Reynolds numbers might collectively contribute substantially to vertical mixing. Scaling analysis indicates that the mixing efficiency η, or the ratio between the rate of potential energy conversion and total work done on the fluid, should scale with $\eta \sim (a/\ell)^3$ as $a/\ell \to 0$, where a is the size of the organism and $\ell = \left(\nu\kappa/N^2\right)^{1/4}$ is an intrinsic length scale of a stratified fluid with kinematic viscosity ν, tracer diffusivity κ, and buoyancy frequency N^2. A regularized singularity model demonstrates this scaling, indicating that in this same limit $\eta \approx 1.2\,(a/\ell)^3$ for vertical swimming and $\eta \approx 0.14\,(a/\ell)^3$ for horizontal swimming. The model further predicts the absolute maximum mixing efficiency of an ensemble of randomly oriented organisms is around 6% and that the greatest mixing efficiencies in the ocean (in regions of strong salt-stratification) are closer to 0.1%, implying that the total contribution of microorganisms to vertical ocean mixing is negligible.

9:18AM M36.00007 Turbulence structure of gravity and turbidity currents , SENTHIL RADHAKRISHNAN, MARIO SCHILLER, ECKART MEIBURG, UC Santa Barbara — DNS of moderate Reynolds number gravity currents has shown that the vortices in the near-wall region are qualitatively similar to the vortices found in the near-wall region of boundary layers, and the Kelvin-Helmholtz billows at the interface between the high and the low density fluid are similar to the ones observed in mixing layers. In the current work, we perform a quantitative study of the turbulence structure of gravity and turbidity currents at higher Reynolds number using LES. In gravity currents, the streamwise Reynolds stresses in the near-wall region match the ones observed in boundary layers. Wall-normal Reynolds stresses are, however, damped due to stable stratification. In turbidity currents, the bulk density in the near-wall region increases due to particle settling, which results in increased damping of Reynolds stresses due to stronger stratification effects. In the interfacial region, the wall-normal Reynolds stresses in gravity currents are damped as compared to the ones observed in mixing layer. In turbidity currents, however, the Reynolds stresses in the interfacial region match the ones in mixing layers. This is a result of weaker stratification as the density difference across the interfacial region decreases due to particle settling closer to the wall.

Tuesday, November 26, 2013 10:30AM - 11:05AM –

Session N27 Invited Session: Transverse Jet Shear Layer Instabilities and Their Control

Spirit of Pittsburgh Ballroom A - J. Philip Drummond, NASA Langley Research Center

10:30AM N27.00001 Transverse jet shear layer instabilities and their control , ANN KARAGOZIAN, University of California, Los Angeles — The jet in crossflow, or transverse jet, is a canonical flowfield that has relevance to engineering systems ranging from dilution jets and film cooling for gas turbine engines to thrust vector control and fuel injection in high speed aerospace vehicles to environmental control of effluent from chimney and smokestack plumes. Over the years, our UCLA Energy and Propulsion Research Lab's studies on this flowfield have focused on the dynamics of the vorticity associated with equidensity and variable density jets in crossflow, including the stability characteristics of the jet's upstream shear layer. A range of different experimental diagnostics have been used to study the jet's upstream shear layer, whereby a transition from convectively unstable behavior at high jet-to-crossflow momentum flux ratios to absolutely unstable flow at low momentum flux and/or density ratios is identified. These differences in shear layer stability characteristics have a profound effect on how one employs external excitation to control jet penetration, spread, and mixing, depending on the flow regime and specific engineering application. These control strategies, and challenges for future research directions, will be identified in this presentation.

Tuesday, November 26, 2013 10:30AM - 11:05AM –

Session N28 Invited Session: Swimming and Running through Sand: Resistive Force Theory in Granular Media

Spirit of Pittsburgh Ballroom B/C - Harry L. Swinney, University of Texas at Austin

10:30AM N28.00001 Swimming and running through sand: resistive force theory in granular media[1] , DANIEL GOLDMAN, Georgia Tech — Resistive force theory (RFT) is often used to analyze the movement of microscopic organisms swimming in fluids. In RFT, a body is partitioned into infinitesimal segments, each of which generates thrust and experiences drag. Linear superposition of forces from elements over the body allows prediction of swimming kinematics and kinetics. While RFT does not always show quantitative agreement with experimental measurements in fluids [e.g. Rodenborn et al, PNAS, 2013], we show that it quantitatively models the locomotion of animals and robots that move on and within dry granular media. RFT shows excellent agreement when the medium is slightly polydisperse, in the regime where frictional forces dominate material inertial forces, and when locomotion can be approximated as confined to a plane. Within a given plane (horizontal or vertical) relationships that govern the force versus orientation of an elemental intruder are functionally independent of the granular medium. We use RFT to explain features of locomotion–these include muscle activation patterns during sand-swimming by the sandfish lizard and optimum limb shape for legged robot walking.

[1]Work supported by NSF and ARL.

Tuesday, November 26, 2013 11:10AM - 11:30AM –

Session P27 Invited Session: Andreas Acrivos Dissertation Award Talk: Turbulence and Internal Waves in Tidal Flow over Topography

Spirit of Pittsburgh Ballroom A - Jonathan Rothstein, University of Massachusetts at Amherst

11:10AM P27.00001 Andreas Acrivos Dissertation Award Talk: Turbulence and Internal Waves in Tidal Flow over Topography , BISHAKHDATTA GAYEN, The Australian National University — Energetic internal waves, commonly known as internal tides, are generated in the ocean by tidal flow interactions with bottom topography like seamounts, ridges, slopes and canyons. In my talk, I will highlight the dynamical processes underlying the turbulence formed during the generation of internal tides at topography and their subsequent interaction with a realistically-stratified upper ocean. The processes have been investigated and quantified using Direct Numerical Simulation (DNS) and Large Eddy Simulation (LES). The focus of the talk will be on ocean topography where, despite gentle barotropic tides, strong near-bottom turbulence has been observed at certain locations. I will present the hypothesis that hotspots of turbulence occur at near-critical locations where the slope angle is nearly equal to the propagation angle. The simulations show transition to turbulence along the entire extent of the near-critical region of the slope when the Reynolds number is of the order 150. The transition is found to be initiated by a convective instability which is closely followed by shear instability. The peak value of the near-bottom velocity is found to increase with increasing length of the critical region of the topography. The scaling law that is observed to link the near-bottom peak velocity to slope length could be explained by a turbulent boundary layer analysis. Maximum turbulent kinetic energy and dissipation rate are found just after the zero velocity point when flow reverses from downslope to upslope motion and the local mean shear is almost zero. The phasing and other characteristics of the turbulent mixing in the present simulations show remarkable similarity with those observed off Kaena Ridge in Hawaii taken during the Hawaiian Ocean Mixing Experiment (HOME), and may be explained by the beam-scale convective overturns found here. In the last part of my talk, I will discuss the interaction between an internal wave beam that is launched from bottom topography and an upper ocean pycnocline, and I will also present the characterization of the cascade to small scales in the context of internal wave beam degradation that is observed in the ocean.

Tuesday, November 26, 2013 11:10AM - 11:30AM –

Session P28 Invited Session: Francois N. Frenkiel Award Talk: Shock Structure in Shock-Turbulence Interactions Spirit of Pittsburgh Ballroom B/C - Malcolm J. Andrews, Los Alamos National Laboratory

11:10AM P28.00001 Francois N. Frenkiel Award Talk: Shock Structure in Shock-Turbulence Interactions[1] , DIEGO DONZIS, Texas A&M University — In many natural and engineering flows, turbulence often interacts with shock waves. Significant efforts have been devoted to understand the effects of the shock on the turbulence in the canonical configuration of turbulence convected through a stationary shock, at a convective Mach number M. Most studies, however, treated the shock as a discontinuity leading to dependencies only on M. However, numerical and experimental evidence shows systematic dependences on Reynolds (R_λ) and turbulent Mach numbers (M_t), the other two non-dimensional parameters in the problem. Even more limited is the understanding of the effect of turbulence on the shock, especially when the shock cannot be assumed to be a discontinuity. This is the main focus of this work. We use general principles of similarity scaling show that consistency with known physical limiting behavior requires incomplete similarity solutions where the governing non-dimensional parameters (R_λ, M and M_t) can be combined to reduce the number of similarity parameters that describes the phenomenon. An important parameter is found to be $K = M_t/R_\lambda^{1/2}(M-1)$ which is proportional to the ratio of laminar shock thickness to the Kolmogorov length scale. The shock thickness under turbulent conditions, on the other hand, is essentially a random variable. Under a quasi-equilibrium assumption, shown to be valid when $K^2 \ll 1$, analytical results are obtained for statistics of the turbulent shock thickness, velocity gradient, and dilatation at the shock. It is shown that these quantities exhibit universal behavior in the parameter K with corrections in $M_t/(M-1)$, for velocity fields with arbitrary statistics. Excellent agreement is observed with available data from direct numerical simulations. We further use the results to understand amplification factors of the streamwise velocity component as well as to determine whether the interaction is in the so-called wrinkled or broken regime.

[1]Support from NSF and AFOSR is gratefully acknowledged.

Tuesday, November 26, 2013 1:05PM - 3:28PM –

Session R1 Geophysical: General III - Open Channels and Sedimentation 323 - Jorge Abad, University of Pittsburgh

1:05PM R1.00001 Vortex dynamics of rectangular lateral cavities in open channel flows: Effects of the aspect ratio on mass transport and residence times[1] , CRISTIAN ESCAURIAZA, KARINA SOTO, CHRISTIAN GONZALEZ, Pontificia Universidad Catolica de Chile, CAI WEI, EMMANUEL MIGNOT, NICOLAS RIVIERE, Laboratoire de mecanique des fluides et d'acoustique, INSA de Lyon, France — Turbulent flows past lateral cavities in rivers and open channels play an important role in many environmental and geophysical applications. Large-scale coherent structures produced in the cavity are the most important mechanisms that control the dispersion and transport of contaminants in streams with transient storage zones. In this work we study the recirculating flow in lateral rectangular cavities with aspect ratios 1.0 and 3.0. We focus on the topology of the flow within the cavity and the dynamics of the shear-layer that forms in the main channel in shallow subcritical flows, using time-resolved Particle Image Velocimetry (PIV) and numerical simulations with coherent-structure resolving turbulent models (DES-LR). Through this investigation we provide new insights on the mechanisms of dispersion and transport of contaminants for each aspect ratio, and analyze the statistics of the mass exchange and residence times in the cavity for both configurations.

[1]supported by Fondecyt 1130940 and ECOS/Conicyt C11E02

1:18PM R1.00002 Hydraulic jumps with upstream shear , KELLY OGDEN, Physical Oceanography, MIT/WHOI, KARL HELFRICH, Physical Oceanography, WHOI — Hydraulic jumps in flows with background shear are investigated, motivated by applications such as the flow over sills in Knight Inlet and the Pre-Bosphorus Channel. The full solution space and allowable solutions to several two-layer theories for hydraulic jumps with upstream shear are identified. The two-layer theories considered, including a recent theory by Borden et al. (JFM, 2012), are distinguished by how dissipation is partitioned between the layers. It is found that upstream shear with a faster and thinner lower layer causes an increase in bore speed, for a given jump height. Further, these two-layer solutions only exist for a limited range of upstream shear. 2D numerical simulations are conducted, guided by the two-layer theory solution space, and the results are compared to the theories. The simulations show the qualitative types of hydraulic transitions that occur, including undular bores, fully turbulent jumps, and conjugate state-like solutions; the type depends on the jump height and upstream shear for fixed upstream layer depths. Numerical simulations are used to investigate the mixing. Finally, a few 3D numerical simulations were made and are found to be consistent with the 2D results.

1:31PM R1.00003 Large-eddy simulation of density currents on inclined beds[1] , SAURABH CHAWDHARY, ALI KHOSRONEJAD, St. Anthony Falls Laboratory, University of Minnesota, Minneapolis, USA, GEORGE CHRISTODOULOU, Department of Civil Engineering, National Technical University of Athens, Athens, Greece, FOTIS SOTIROPOULOS, St. Anthony Falls Laboratory, University of Minnesota, Minneapolis, USA — Density currents are stratified flow in presence of density differential and gravity field. We carry out Large-Eddy Simulation (LES) to simulate the flow of a density current formed over sloped bed due to an incoming jet of heavy density salty water for two different cases of bed slope: (a) 5 degrees and (b) 15 degrees. The Reynolds and Richardson numbers based on inlet height and inlet velocity were (a) 1100 and 0.471, and (b) 2000 and 0.0355, respectively. The Schmidt number is set equal to 620, which corresponds to the value for salt-water. The computed results are compared with laboratory experiments in terms of overall shape of the heavy-density plume and its spreading rate and are shown to be in reasonable agreement. The instantaneous LES flow fields are further analyzed to gain novel insights into the rich dynamics of coherent vortical structures in the flow. The half-width of the plume is plotted as a function of downstream length and found to exhibit three different regions on a log scale, in agreement with previous experimental findings.

[1]We acknowledge computational support from the Minnesota Supercomputing Institute.

1:44PM R1.00004 Large-eddy simulation of coupled turbulence, free surface, and sand wave evolution in an open channel , ALI KHOSRONEJAD, FOTIS SOTIROPOULOS, University of Minnesota, ST. ANTHONY FALLS LAB TEAM — We develop and validate a coupled 3D numerical model for carrying out three-phase large-eddy simulations of turbulence, free-surface, and sand waves-bed morphodynamics under live bed conditions. We employ the Fluid-Structure Interaction Curvilinear Immersed Boundary (CURVIB) method of Khosronejad et al. (Adv. in Wat. Res.,2011). The LES is implemented in the context of the CURVIB method using wall modeling (Kang and Sotiropoulos, Adv. in Wat. Res.,2011). Free-surface motion is simulated by coupling the CURVIB method with a two-phase level set approach as in Kang and Sotiropoulos (Adv. in Wat. Res.,2012). Transport of bed load and suspended load sediments are combined in the non-equilibrium form of the Exner for the bed surface elevation, which evolves due to the spatio-temporally varying bed shear stress field induced by the turbulent flow. Simulations are carried out for the experiments of Venditti et al. (2005). It is shown that the model can accurately capture sand-wave initiation, growth, and migration processes observed in the experiment. The effects of free-surface on bed-form dynamics is also quantified by comparing the three-phase simulation results with two-phase simulations using a fixed rigid-lid as the free surface. This work is supported by NSF Grants EAR-0120914 and EAR-0738726, and National Cooperative Highway Research Program Grant NCHRP-HR 24-33.

1:57PM R1.00005 A generalized shallow-water analysis of gravity currents in various cross-area channels for Boussinesq and non-Boussinesq systems , MARIUS UNGARISH, Technion, Haifa — The propagation of a high-Reynolds-number gravity current in a horizontal channel along the horizontal coordinate x is considered. The bottom and top of the channel are at $z = 0, H$, and the cross-section is given by the quite general $-f_1(z) \le y \le f_2(z)$ for $0 \le z \le H$. A shallow-water formulation is presented and used for the solution of the dam-break problem. The dependent variables are the position of the interface, $h(x,t)$, and the speed (averaged over the area of the current), $u(x,t)$. The non-rectangular cross-section enters the formulation via $f(h)$ and integrals of $f(z)$ and $zf(z)$, where $f(z) = f_1(z) + f_2(z)$ is the width of the channel. For a given geometry $f(z)$, the input parameters in the lock-release problem are the ratios of height H/h_0 and density ρ_a/ρ_c, of ambient to lock fluids. The dam-break problem can be solved by the method of characteristics, but complications (jumps, critical restrictions) appear when the return flow in the ambient is significant; these features are not captured by a one-layer model, and hence a two-layer model solution is introduced. A strong generalization is achieved: the standard classical solutions for a rectangular or unbounded channel are particular cases of the present theory.

2:10PM R1.00006 Flow over interacting barchan dunes studied in a refractive-index-matched environment , Z. TANG, N. JIANG, Tianjin University, G. BLOIS, J.M. BARROS, J.L. BEST, K.T. CHRISTENSEN, Univ. of Illinois — Barchan dunes are three-dimensional topographic features characterized by a crescentic shape. Very common on Earth's surface, barchans are produced by unidirectional flows in regions of sediment starvation and are characterized by migration rates that are a function of their volume. This results in complex dune-to-dune interaction mechanisms that are poorly understood. In order to quantify the flow structure produced by interacting barchans, PIV measurements were made wherein the dune models were immersed in a flowing fluid that was refractive-index-matched to the dune material. Doing so provided full optical access to the obstructed regions of flow and eliminated reflections from the liquid-solid boundaries, allowing near-wall data to be collected. Clear barchan models with different volumetric ratios were arranged in tandem, and flow-field measurements were made in multiple streamwise–wall-normal and streamwise–spanwise planes. Ensemble-averaged flow fields and Reynolds stresses were obtained for different barchan spacings and compared to the reference case of an isolated barchan. Proper orthogonal decomposition analysis was employed to study the spatial characteristics of the energy distribution both between and downstream of the aligned dunes.

2:23PM R1.00007 Bedforms migration and interactions: a PIV investigation , G. BLOIS, J.L. BEST, J.M. BARROS, K.T. CHRISTENSEN, Univ. of Illinois — Bedforms, such as ripples and dunes, are ubiquitous in natural environments in which solid particles are immersed in a moving fluid that is above the critical bed shear stress for sediment movement. The mutual interactions between flow and bed topography result in processes in which both flow and bed morphology are unsteady and dynamic. Bedforms with different sizes, shapes and migration rates produce bedform superimposition and amalgamation whose marks are left in the rock record as a specific stratigraphic signature. We investigate the flow associated with amalgamating mobile bedforms using a narrow (5 mm width) flume coupled with PIV, which allows the behavior of quasi two-dimensional bedforms to be observed and quantified. Simultaneous measurements of both the morphology and flow during amalgamation provide a tool to assess validity of current theory and shed new light on the physics of this fundamental problem. Shear layer interactions between adjacent bedforms, leeside erosion and downstream bedform stalling due to the sheltering effect of an upstream bedform are found to be the key aspects of the amalgamation process. The implications of these processes with respect to flow resistance and transition between bedform states are discussed.

2:36PM R1.00008 Suspension and transport of sediment under a plunging wave breaker , XINHUA LU, State Key Laboratory of Water Resources and Hydropower Engineering Science, Wuhan University, XIN GUO, Department of Mechanical Engineering & St. Anthony Falls Laboratory University of Minnesota, Twin Cities, YI LIU, American Bureau of Shipping, LIAN SHEN, Department of Mechanical Engineering & St. Anthony Falls Laboratory University of Minnesota, Twin Cities — To understand the mechanism of suspension and transport of sediment under breaking water waves, we perform large-eddy simulations of a plunging breaker over seabed. The breaking water surface is captured by a coupled level-set and volume-of-fluid method. The mass exchange of sediment between the water region and the bottom is computed through the local upward erosion and downward deposition fluxes. The erosion flux is modeled based on the local shear stress at the bottom, and the deposition flux is estimated based on the sediment concentration near the bottom. We analyze in detail the instantaneous velocity and sediment concentration fields, the erosion and deposition fluxes near the bottom, as well as the bottom deformation under breaking waves. It is found that the sediment is mainly picked up from the bottom at the early stage of wave breaking, brought upwards, mixed by the turbulent motion, and then transported in the wave propagation direction by the current generated by the breaker. The wave breaking significantly enhances the horizontal transport of the sediment. It is also found that the air pocket entrained by the breaking wave plays an important role in the suspension, transport, and redistribution of sediment.

2:49PM R1.00009 Investigation of the mechanism of contaminant release through the sediment-overlying water interface[1] , JIA-HONG GUO, SHU-JUN ZHENG, DAO-ZENG WANG, Shanghai Institute of Applied Mathematics and Mechanics, Shanghai University, ENVIRONMENT FLUID DYNAMICS COLLABORATION — After the external pollutant discharge has been reduced, the release of the contaminant from the sediment to the overlying water may cause the river and lake be contaminated again. On the condition that the overlying water flow does not lead to sediment suspension, numerical and experimental researches are carried out for the contaminant release mechanism through the sediment-overlying water interface. In the numerical simulation, the overlying water flow is calculated as turbulent flow. The sediment is regarded as isotropic homogeneous porous medium, therefore the seepage field in the porous sediment layer is obtained by solving Darcy's equations. Several coupled two dimensional steady and unsteady flows of the overlying water and the pore water in the sediment are calculated. Based on the flow fields obtained, the unsteady contaminant solute transportation process in the sediment and the overlying water is numerically simulated, as the shapes of the sediment-overlying water interface are flat or periodic triangular respectively. The numerical results agree with the experimental results quite well. The results show that the exchange of the pore water and the overlying water is an important factor which decides the release flux of the contaminant from the sediment to the overlying water.

[1]Project supported by the National Natural Science Foundation of China (Grant No. 11032007) and Shanghai Program for Innovative Research Team in Universities.

3:02PM R1.00010 Modeling of matrix acidizing process under reservoir conditions[1] , KARLYGASH TUREGELDIEVA, BAKHYTZHAN ASSILBEKOV, UZAK ZHAPBASBAYEV, Kazakh-British Technical University, ANATOLY ZOLOTUKHIN, Gubkin Russian State University of oil and gas, TIMUR BEKIBAEV, NURLAN KENZHEBEKOV, Kazakh-British Technical University, GUBKIN RUSSIAN STATE UNIVERSITY OF OIL AND GAS COLLABORATION — Effectiveness of the process depends on the parameters: well choice, geological structure of the reservoir, definition of physical and chemical properties of rocks and fluids, agent choice. There are different mathematical models of the matrix acidizing, including the two scale model. These models describe the process in the core scale and Darcy scale, i.e. in an area with dimensions of several centimeters. It leads to the main problem – how to use these models to the near wellbore scale under reservoir conditions. Some authors have increased the dimensions of the cores in numerical simulations and investigated the influence of the core dimensions to acidizing process. In this paper effort to indirectly solve this problem made. It based on boundary conditions alteration and simultaneous solution of matrix acidizing in damaged zone and reservoir fluid flow models. Furthermore in this work the criterion of the acid injection shut down for optimal breakthrough volume calculation was modified. Influence of boundary conditions on near well-bore zone treatment process was investigated.

[1]Science Committee of Ministry of Education and Science of Republic of Kazakhstan

3:15PM R1.00011 Turbulent flow structure during the amalgamation process of river bed forms , CHRISTIAN FRIAS, JORGE ABAD, University of Pittsburgh — Most of the fluvial channels present bedforms such as dunes and ripples which are product of the interaction between hydrodynamics and sediment transport. Although, the effect of this bedforms has been studied by several researchers, very little has been done to study the ripple-dune transition. The purpose of the present study is to extend the observations made by previous experimental results on the bedforms amalgamation process. To this end, three LES cases were carried out to replicate an amalgamation process, a train of ripples (RUN I), a superimposed bed forms (RUN II) and a complete amalgamated bed forms stage(RUN III). Although the experimental base case involved two dimensional bedforms, a three dimensional simulation was developed to analyze the effects of superimposition in transversal and streamwise directions. Some important conclusions from this study are: the region of high shear stresses was related to turbulence production, in which the streamwise velocity fluctuations were associated to the modification of the bed morphology. The turbulence Horseshoes Vortices (THV) were more frequent in RUN I than in the other two cases. Finally, the frequency of the bursting events increased from RUN I to RUN II and decreased from RUN II to RUN III.

Tuesday, November 26, 2013 1:05PM - 3:15PM –

Session R2 Convection and Buoyancy-Driven Flows VII: Gravitational Effects and Flows Past Moving Bodies 324 - Henry Burridge, University of Cambridge

1:05PM R2.00001 Effect of single silica gel particle adsorption on the transport processes in a humid air stream , APRATIM SANYAL, SAPTARSHI BASU, PRAMOD KUMAR, Indian Institute of Science, Bangalore — The effect of adsorption due to a single silica gel particle on a convective field consisting of humid air has been investigated numerically. The adsorption is incorporated as a sink term in the transport equation for species (water vapor) and has been modeled using Linear Driving Force model, while the heat released due to adsorption is taken as source term in the energy equation and proportional to the amount of water vapor adsorbed. The heat released creates a coupling between the species and the temperature field as the adsorption characteristics are directly influenced by particle temperature. The extent of species and temperature boundary layer show the diffusion of the adsorption effects into the free stream. Surface adsorption is found to decrease with Reynolds no. The particle surface temperature increases from forward stagnation point till downstream. This work provides a model for understanding the adsorption kinetics in convective stream for other adsorbate-adsorbent pair. Further more complex scenarios can be modeled such as presence of multiple adsorbent particles, the interaction of species and temperature boundary layers setup due to individual particles and their influence on the overall adsorption characteristics.

1:18PM R2.00002 The effect of noncondensables on the thermocapillary-buoyancy convection in volatile fluids , TONGRAN QIN, LAURA ANFINSON, George W. Woodruff School of Mechanical Engineering, Georgia Institute of Technology, Atlanta, GA, ROMAN GRIGORIEV, School of Physics, Georgia Institute of Technology, Atlanta, GA — Convection in a liquid layer with free surface subject to a horizontal temperature gradient is one of the classic problems in fluid mechanics and heat transfer. In thicker layers the flow is driven by a combination of buoyancy and thermocapillarity in the liquid layer. Unlike buoyancy, thermocapillarity depends rather sensitively on the boundary conditions at the liquid-vapor interface. In particular, for volatile fluids, convective patterns are found to vary significantly with the composition of the gas phase. The interfacial temperature which defines thermocapillary stresses is controlled by the variation of the vapor concentration along the interface. At ambient conditions when the gas phase is dominated by air, the latter is found to be controlled by diffusion of vapor in (essentially static) air. At the opposite extreme where vapor dominates, we find that it is the diffusion of air in (quickly moving) vapor that controls the concentration and temperature distribution, leading to significant differences in the flow patterns found in these two limits. These findings show that the results of the studies conducted under atmospheric conditions are not applicable in the (near) absence of noncondensables (air) and resolve the disagreement between previous numerical and experimental results in the vapor-dominated limit.

1:31PM R2.00003 Gravity driven current during sessile drop coalescence on a surface , YING ZHANG, SAMUEL OBERDICK, STEPHEN GAROFF, Physics Department, Carnegie Mellon University, SHELLEY ANNA, Chemical Engineering Department and Mechanical Engineering Department, Carnegie Mellon University — We study the mixing behavior of two sessile drops following coalescence on a flat surface. The surface is composed of silicone elastomer on which the drops exhibit contact angles of about 90 degree. The two drops are of equal volume at coalescence, but different densities and viscosities. Using laser induced fluorescence, we obtain both a top view of the contact line motion and a side view of the cross-sectional flow. During the coalescence stage, the initial healing of the meniscus bridge and damping of capillary waves occur on time scales comparable to the inertio-capillary time. However, the interface between the dyed and undyed components remains sharp, with diffusive mixing occurring at much longer timescales. At intermediate time scales the motion is controlled by a gravity current, which leads to the eventual stratification into two separate horizontal layers within the composite drop. Using lubrication analysis, we characterize the gravity current as a function of the drop sizes, and the density and viscosity differences between the two merging fluids. The numerical solution of the lubrication analysis captures the observed scaling of the time dependent interface movement as a function of fluid and geometric parameters.

1:44PM R2.00004 Long-Lasting Effect of Initial Configuration in Gravitational Spreading of Material Fronts[1] , NADIM ZGHEIB, Department of Mechanical and Aerospace Engineering, University of Florida, Gainesville, FL 32611, USA, THOMAS BONOMETTI, Université de Toulouse; INPT, UPS; IMFT; UMR 552, Allée Camille Soula, F-31400 Toulouse, France, S. BALACHANDAR, Department of Mechanical and Aerospace Engineering, University of Florida, Gainesville, FL 32611, USA — We present results from experiments and simulations pertaining to finite release gravity currents with a non-axisymmetric cross-section. First, we demonstrate that, contrary to expectation, the effects of the initial shape strongly influence the current's evolution well into the self-similar phases. Then we identify the physical mechanisms responsible for this dependence and propose a simple model capable of well capturing the dynamics of such releases. Finally, we show that this dependence on initial configuration is robust for various types of gravity currents (homogeneous and inhomogeneous) over a wide range of parameters such as Reynolds number, density ratio, wall friction and aspect ratio. We additionally inspect the deposition patterns pertaining to the abovementioned inhomogeneous currents.

[1]Chateaubriand Fellowship; NSF OISE-0968313

1:57PM R2.00005 Fountain behaviour from the frequency of fountain-top fluctuation and rise height , HENRY BURRIDGE, GARY HUNT, University of Cambridge — Focusing on the dynamics of established turbulent axisymmetric fountains in uniform quiescent environments, we present measurements for the frequency of fluctuations at the top of aqueous-saline fountains. Our results span source Froude numbers, $0.3 < Fr_0 < 40$, and clearly indicate four discrete bands of Fr_0 within which the dimensionless frequency of fluctuation is constant - the value of the constant changing between each band [Burridge & Hunt (2013), J. Fluid Mech. 728, 91-119]. Our independent measurements of rise height [Burridge & Hunt (2012),J. Fluid Mech. 691, 392-416] show that established fountains can be classified into four bands - within each, the mean rise height exhibits a unique dependence on Fr_0. Comparison of the Fr_0 bands show close agreement. This suggests that the dominant physics controlling the fountain's behaviour remains unchanged within each band and that monitoring the frequency of fluctuations at the fountain top provides a robust and independent means of classification. Within each class we identify unique time scales for the fluctuations and decompose these time scales into the relevant rise height and velocity scales, thereby demonstrating that the dominant length, time and velocity scales all change at the same Fr_0 boundaries.

2:10PM R2.00006 Dynamic Separation on a Pitching and Surging Airfoil as a Model for Flow over Vertical Axis Wind Turbine Blades[1] , REEVE DUNNE, BEVERLEY MCKEON, California Institute of Technology — The flow over a pitching and surging NACA 0018 airfoil at a chord Reynolds number of 100,000 is investigated using 2D time resolved particle image velocimetry. Sinusoidal pitch experiments between $\pm 30^\circ$ at a reduced frequency $\mathrm{k} = \frac{\Omega \mathrm{c}}{2\mathrm{U}_\infty} = .12$ closely simulate the unsteady angle of attack experienced by the blade of a representative commercial vertical axis wind turbine (VAWT). The unsteady separation process is analyzed in detail with a focus on development of vorticity at the leading edge. Reduced order modeling techniques are used to deconstruct the flow and identify the evolution of dominant flow structures over the pitching cycle. Surging at the same reduced frequency and $\frac{\mathrm{U_{max}} - \mathrm{U_{min}}}{\mathrm{U_{mean}}} = .80$ is added to investigate the effect of the Reynolds number variation associated with the rotation of a VAWT blade in a non-rotating, laboratory frame.

[1]This research is funded by the Gordon and Betty Moore Foundation through Grant GBMF #2645 to the California Institute of Technology.

2:23PM R2.00007 Coriolis effect on dynamic stall in a vertical axis wind turbine[1] , HSIEH-CHEN TSAI, TIM COLONIUS, California Institute of Technology — The immersed boundary method is used to simulate the flow around a two-dimensional rotating NACA 0018 airfoil at moderate (sub-scale) Reynolds number in order to investigate separated flow occurring on a vertical-axis wind turbine (VAWT). The influence of dynamic stall on the forces is characterized as a function of tip-speed ratio. The influence of the Coriolis effect is also investigated by comparing the rotating airfoil to one undergoing a surging and pitching motion that produces an equivalent speed and angle-of-attack variation over the cycle. While the Coriolis force produces only small differences in the averaged forces, it plays an important role during dynamic stall. Due to the fact that the Coriolis force deflects the fluid and propagates the vortices differently, the wake-capturing phenomenon of the trailing edge vortex is observed in the flow around the rotating airfoil during a certain range of azimuthal angle. This wake-capturing of the trailing edge vortex leads to a large decrease in lift. However, because of the phase difference between each wake-capturing, there are only small differences in the average forces. The simulations are also compared to results from companion water-tunnel experiments at Caltech.

[1]This project is supported by the Gordon and Betty Moore Foundation

2:36PM R2.00008 Unsteady vortex dynamics for finite-aspect-ratio pitching wings[1] , RYAN JANTZEN, KUNIHIKO TAIRA, Florida State University, KENNETH GRANLUND, MICHAEL OL, U.S. Air Force Research Laboratory — We examine the vortex dynamics around low-aspect-ratio pitching wings and the corresponding unsteady aerodynamic loading using direct numerical simulations with the immersed boundary projection method. Finite-aspect-ratio wings of two and four with rectangular planforms are considered to pitch about the leading edge, from an angle of attack of $\alpha = 0$ to 45°, over a range of reduced frequencies for a Reynolds number of 300. These parameters are chosen to uncover the influence of fast and slow wing motion on the formation of the wake vortices. We also study the relationship between the vortex dynamics and the unsteady forces exerted on the pitching wing emphasizing the influence from the leading-edge and tip vortices. Companion water tunnel experiments are performed with fluorescent dye visualization and direct force measurements at a Reynolds number of $20,000$, which highlight qualitative similarities between the flow fields despite the large difference in Reynolds numbers. The insights obtained from the numerical and experimental results are used as a foundation for the development of closed-form models to predict aerodynamic forces for wings undergoing large-amplitude maneuvers representative of flapping wing kinematics and wings in gusty operating conditions.

[1]Work supported by the ASEE Summer Faculty Fellowship Program.

2:49PM R2.00009 Force Coefficients on Surging Rigid and Flexible Wings , PETER MANCINI, ANYA JONES, University of Maryland, KENNETH GRANLUND, MICHAEL OL, Air Force Research Laboratory — This study considers an aspect ratio 4 rigid flat plate and an aspect ratio 4.5 flexible wing, undergoing rectilinear motion in a water tunnel over several chord lengths at a Reynolds number of 20,000. Varying incidence angle, Reynolds number, and acceleration profile led to an extensive parameter study for both wings. Acceleration regions were linear with time and varied with distances of 0.25 to 6.0 chord-lengths. Measurements include lift and drag histories along with flow visualization of leading and trailing edge vortices throughout the entire motion by fluorescent dye injection illuminated by a laser sheet. A non-circulatory bump in lift coefficient at the end of the acceleration region was observed for each rigid wing case. The rigid wing also experienced a significant decrease in lift shortly after the wing reached its terminal velocity. This dip was followed by a second peak in lift around 6 chords traveled for every case, although the magnitudes differed among the acceleration profiles. Conversely, the flexible wing exhibited little to no non-circulatory peak at the end of acceleration and did not experience this dip and rise in lift. This study explores the influence of planform and chordwise flexibility on leading edge vortex formation, retention, and shedding.

3:02PM R2.00010 Unsteady pitching flat plates , KENNETH GRANLUND, MICHAEL OL, Air Force Research Laboratory, LUIS BERNAL, University of Michigan — Direct force measurements and qualitative flow visualization were used to compare flowfield evolution vs. lift and drag for a nominally 2D rigid flat plate executing smoothed linear pitch ramp maneuvers in a water tunnel. Non-dimensional pitch rate was varied from 0.01 to 0.5, incidence angle from 0 to 90 degrees, and pitch pivot point from the leading to the trailing edge. For low pitch rates, the main unsteady effect is delay of stall beyond the steady incidence angle. Shifting the time-base to account for different pivot points leads to collapse of both lift/drag history and flowfield history. For higher rates, a leading edge vortex forms; its history also depends on pitch pivot point, but linear shift in time-base is not successful in collapsing lift/drag history. Instead, a phenomenological algebraic relation, valid at the higher pitch-rates, accounts for lift and drag for different rates and pivot points, through at least 45 degree incidence angle.

Tuesday, November 26, 2013 1:05PM - 3:15PM –

Session R3 Multiphase Flows VIII 325 - Anthony Rosato, New Jersey Institute of Technology

1:05PM R3.00001 First particle acceleration measurements for a shocked multiphase flow at a new horizontal shock tube facility , GREG ORLICZ, ADAM MARTINEZ, KATHY PRESTRIDGE, Los Alamos National Laboratory, EXTREME FLUIDS TEAM — The horizontal shock tube at Los Alamos, used for over 20 years to study shock-driven mixing between different density gases, has been retrofitted with a new particle seeding system, test section, and diaphragmless driver to investigate the unsteady forces on particles as they are accelerated by a shock wave. Current experiments are performed to measure the acceleration of dispersed glycol droplets, with nominal 0.5 μm diameter, carried in ambient air. Measurements at this facility will be used to develop and validate empirical models implemented in numerical codes. A Particle Image Velocimetry/Accelerometry (PIVA) system is implemented at the facility using eight laser pulses and an eight-frame high speed camera. The lasers are 532 nm Nd:YAGs with pulse widths of 20 ns, and the camera is a Specialised Imaging SIMD with 1280x960 resolution at up to 7 million frames per second. With this PIVA arrangement, eight particle fields are collected by independently varying the interframe times. Seven velocity and six acceleration fields are used to study the unsteady drag on the particles. Initial data sets are with a size distribution of known particle diameters. Plans are to vary the particle/gas density ratio, particle diameters, and particle phase (liquid/solid).

1:18PM R3.00002 Measurements of Multiphase Fluid Mixing Using Synchrotron X-Ray Fluorescence[1] , ALAN KASTENGREN, Advanced Photon Source, Argonne National Laboratory, BENJAMIN HALLS, TERRY MEYER, Iowa State University — Multiphase flows can prove problematic for the use of optical diagnostics due to the strong interaction of visible light with phase boundaries. X-ray absorption and phase-contrast imaging have been successfully used to probe multiphase fluid flows under a wide variety of conditions. This presentation will describe the use of another technique, x-ray fluorescence spectroscopy, to probe an impinging jet spray flowfield. The x-ray fluorescence technique will be described, including its advantages and drawbacks compared to other techniques, both optical and x-ray. Preliminary results from the impinging jet flowfield show that the fluid from each initial jet tends to congregate on the side of the sheet formed after the impingement point opposite the jet. This behavior was not expected prior to these measurements, demonstrating the utility of the fluorescence technique to probe the mixing of the two streams. Other potential applications for the x-ray fluorescence technique will also be briefly discussed.

[1]Use of the Advanced Photon Source, an Office of Science User Facility operated for the U.S. Department of Energy (DOE) Office of Science by Argonne National Laboratory, was supported by the U.S. DOE under Contract No. DE-AC02-06CH11357

1:31PM R3.00003 Role of fluctuations in instability generation in gas-solid suspensions[1] , SHANKAR SUBRAMANIAM, MOHAMMAD MEHRABADI, RAVI KOLAKALURI, SUDHEER TENNETI[2], Iowa State University — Stability analysis of gas-solid suspensions using kinetic theory (Koch, Phys. Fluids, 1990) relies on a number density function (NDF) that is based on the canonical (constant number) ensemble. Euler-Lagrange simulations of a model problem are used to show that this approach does not accurately represent the scale–dependent interphase coupling between different realizations of the gas velocity field and fluctuations in the number of particles naturally occurring in fluidized beds. The grand-canonical (or variable number) ensemble is better suited to representing this coupling, and it is shown how the NDF can be related to this ensemble. The evolution of the grand-canonical NDF then leads to instabilities and growth of spatial fluctuations in the number density of a homogeneous suspension. This analysis leads to a different explanation for the growth of instabilities in homogeneous gas-solid suspensions that does not require perturbations in the average number density. Rather it is shown that the interaction of different realizations of the gas velocity field with individual realizations of the particle field leads to the growth of instabilities due to the dependence of drag on local volume fraction in each realization, that is extracted from particle-resolved DNS data.

[1]This work is partially supported by NSF CBET 1134500
[2]now at CD-Adapco, Lebanon, NH

1:44PM R3.00004 Towards large-eddy simulation of multiphase flows using two-way coupled, Euler-Lagrangian methods , WYATT HORNE, KRISHNAN MAHESH, University of Minnesota — Two-way coupled Euler-Lagrangian methods are sensitive to the size of the particle with respect to the Eulerian grid. We develop an interpolation methodology that addresses this issue for unstructured grids. The carrier fluid is solved using large-eddy simulation (LES) including finite size effects and force coupling from the Lagrangian particles. The Lagrangian particle motion is solved using equations relating the motion of the carrier fluid to forces on each discrete particle. Interpolation of Lagrangian quantities to Eulerian quantities is performed using interpolation kernels dependent on particle size that are volume averaged over control volumes. This interpolation technique is compared to other interpolation methods over several canonical flow cases. It is found from these comparisons that the developed interpolation technique is capable of producing more accurate results. Results are shown for both bubbles and solid particles. Simulations of a single sphere rising in an inclined channel under conditions similar to an experiment conducted by Lomholt et al. [Int. J. Multiphase Flow (2002) **28**:225–246] are performed. Good agreement is found between the experimental and simulated particle trajectories and velocity profiles.

1:57PM R3.00005 Eulerian-Lagrangian Simulations of Bubbly Flows in A Vertical Square Duct[1] , RUI LIU, SURYA P. VANKA, BRIAN G. THOMAS, Department of Mechanical Science and Engineering, University of Illinois at Urbana-Champaign — We report results of Eulerian-Lagrangian simulations of developing upward and downward bubbly flows in a vertical square duct with a bulk Reynolds number of 5000. The continuous fluid is simulated with DNS, solving the Navier-Stokes equations by a second-order accurate finite volume fractional step method. Bubbles of sizes comparable to the Kolmogorov scale are injected at the duct entrance with a mean bulk volume fraction below 10^{-2}. A two-way coupling approach is adopted for the interaction between the continuous fluid phase and dispersed bubble phase. The bubbles are tracked by a Lagrangian method including drag and lift forces due to buoyancy and Saffman lift. A in-house code, CU-FLOW, implemented on Graphic Processing Unit (GPU) is used for simulations in this work. The preferential distributions of bubbles and their impact on local turbulence structures and their effects on turbulent kinetic energy budgets are studied. Results between an upward flow and a downward flow with the bubbles are compared.

[1]Work Supported by Continuous Casting Consortium at UIUC.

2:10PM R3.00006 Low Reynolds-number hydrodynamics of immersed fluid sheets , NEIL RIBE, BINGRUI XU, Lab FAST, CNRS/Univ Paris-Sud, Orsay, France — Low Reynolds-number flows of thin bodies of viscous fluid immersed in an external fluid with a different viscosity occur in contexts ranging from microfluidics to global geophysics. Here we study the buoyancy-driven motion of a two-dimensional sheet with thickness h and viscosity η_2 in a less dense fluid with viscosity η_1, starting from an initial geometry that corresponds to subduction of oceanic lithosphere in Earth's mantle. We work with two different representations of the flow: a full boundary-integral formulation, and a new "hybrid" integral equation that combines asymptotic thin-sheet theory with a boundary-integral representation of the external flow. In both cases, the time-dependent motion of the sheet is obtained by updating the geometry after each instantaneous flow solution. A scaling analysis shows that the sheet's velocity is controlled by its dimensionless "stiffness" $S \equiv (\eta_2/\eta_1)(h/\ell_b)^3$, where the "bending length" ℓ_b is the length of the portion of the sheet's midsurface where bending moments are significant. We will present illustrative simulations of the evolving sheet as a function of the viscosity ratio η_2/η_1, and will assess the relative efficiencies of the full boundary-integral and hybrid approaches.

2:23PM R3.00007 Extension of the Mass-Conserving Level-Set method to unstructured polyhedral control volumes for two-phase flows , FAHIM RAEES, DUNCAN R. VAN DER HEUL, KEES VUIK, Delft University of Technology — In this research, we present the Mass-Conserving Level-Set method (MCLS) for the simulation of two-dimensional, incompressible, immiscible two-phase flows, using a discretisation scheme that can accurately and efficiently handle domains of arbitrary geometrical complexity. The level set and the volume of fluid fraction are evolved at each time step on unstructured triangular grids. The Higher-Order Discontinuous Galerkin finite element method is used for spatial discretisation of the level set advection equation. The volume of fluid fraction advection is done in geometrical manner using Lagrangian-Eulerian method. This method is accurately mass conserving and easy to implement on unstructured grids. Also, it avoids overlapping regions during the volume of fluid fraction advection. The advected level set is corrected locally to make it mass conserving by the means of an explicit, invertible relation between the local level set and the volume of fluid fraction. This relation is termed as a Volume-of-Fluid function. The results show that proposed method is accurately mass conserving. Also, higher-order convergence is highlighted with this method on unstructured grids for the different test cases.

2:36PM R3.00008 Computational simulation of the interactions between water waves and two-dimensional wave energy converters , AMIRMAHDI GHASEMI, ASHISH PATHAK, ROBERT CHIODI, MEHDI RAESSI, University of Massachusetts Dartmouth — Ocean waves represent a vast renewable energy resource, which is mostly untapped. We present a computational tool for simulation of the interactions between waves and two-dimensional oscillating solid bodies representing simple wave energy converters (WECs). The computational tool includes a multiphase flow solver, in which the two-step projection method with GPU acceleration is used to solve the Navier-Stokes equations. The fictitious domain method is used to capture the interactions of a moving rigid solid body with the two-fluid flow. The solid and liquid volumes are tracked using the volume-of-fluid (VOF) method, while the triple points and phase interfaces in three-phase cells are resolved. A consistent mass and momentum transport scheme is used to handle the large density ratio. We present results of two wave generation mechanisms with a piston or flap wave maker, where the theoretical and experimental results were used for validation. Then, simulation results of several simple devices representative of distinct WECs, including a bottom-hinged flap device as well as cylindrical or rectangular terminators are presented. The results are in good agreement with the available experimental data.

2:49PM R3.00009 Simulating immiscible multi-phase flow and wetting with 3D stochastic rotation dynamics (SRD) , THOMAS HILLER, MARTA SANCHEZ DE LA LAMA, STEPHAN HERMINGHAUS, Max Planck Institute for Dynamics and Self-Organization, MARTIN BRINKMANN, Saarland University, Germany — We use a variant of the mesoscopic particle method stochastic rotation dynamics (SRD) to simulate immiscible multi-phase flow on the pore and sub-pore scale in three dimensions. As an extension to the multi-color SRD method, first proposed by Inoue et.al., we present an implementation that accounts for complex wettability on heterogeneous surfaces. In order to demonstrate the versatility of this algorithm, we consider immiscible two-phase flow through a model porous medium (disordered packing of spherical beads) where the substrate exhibits different spatial wetting patterns. We show that these patterns have a significant effect on the interface dynamics. Furthermore, the implementation of angular momentum conservation into the SRD algorithm allows us to extent the applicability of SRD also to micro-fluidic systems. It is now possible to study e.g. the internal flow behaviour of a droplet depending on the driving velocity of the surrounding bulk fluid or the splitting of droplets by an obstacle.

3:02PM R3.00010 Vertical gas injection into liquid cross-stream beneath horizontal surfaces , IN-HO LEE, SIMO MAKIHARJU, University of Michigan, INWON LEE, Pusan National University, MARC PERLIN, STEVE CECCIO, University of Michigan — Skin friction drag reduction on flat bottomed ships and barges can be achieved by creating an air layer immediately beneath the horizontal surface. The simplest way of introducing the gas is through circular orifices; however the dynamics of gas injection into liquid cross-streams under horizontal surfaces is not well understood. Experiments were conducted to investigate the development of the gas topology following its vertical injection through a horizontal surface. The liquid cross-flow, orifice diameter and gas flow rate were varied to investigate the effect of different ratios of momentum fluxes. The testing was performed on a 4.3 m long and 0.73m wide barge model with air injection through a hole in the transparent bottom hull. The incoming boundary layer was measured via a pitot tube. Downstream distance based Reynolds number at the injection location was 5 x 10^5 through 4 x 10^6. To observe the flow topology, still images and video were recorded from above the model (i.e. through the transparent hull), from beneath the bottom facing upward, and from the side at an oblique angle. The transition point of the flow topology was determined and analyzed.

Tuesday, November 26, 2013 1:05PM - 3:15PM –

Session R4 Instability: Boundary Layers I - Surface Topography 326 - Pierre Ricco, The University of Sheffield

1:05PM R4.00001 Distributed Roughness and Transient Growth in a Flat Plate Boundary Layer[1] , MATTHEW KUESTER, EDWARD WHITE, Texas A&M University — Very few experiments or numerical simulations have deliberately studied how distributed surface roughness affects boundary layer transition through the transient-growth mechanism. In this experimental work, transient growth initiated by randomly distributed, streamwise-extended surface roughness is experimentally studied in the Klebanoff–Saric Wind Tunnel at Texas A&M University. Two distributed roughness surfaces with the same surface topography, but different amplitudes, were manufactured and mounted flush with the wall in a flat-plate boundary layer. Detailed measurements of the roughness wake were made using hotwire anemometry at three sub-critical Reynolds numbers. Transient growth was observed at multiple spanwise wavelengths. Multiple wavelengths underwent robust growth but only weak growth was observed near the so-called "optimal" wavelength. Comparing cases with different Reynolds numbers and roughness heights reveals how the steady velocity disturbance scales and gives futher insight into the receptivity mechanisms of distributed surface roughness.

[1]The authors would like to acknowledge the support of the Air Force Office of Scientific Research through AFOSR Grant FA9550-11-1-0203.

1:18PM R4.00002 Appraisal of boundary layer trips for landing gear testing , PHILIP MCCARTHY, GRAHAM FELTHAM, ALIS EKMEKCI, University of Toronto — Dynamic similarity during scaled model testing is difficult to maintain. Forced boundary layer transition via a surface protuberance is a common method used to address this issue, however few guidelines exist for the effective tripping of complex geometries, such as aircraft landing gears. To address this shortcoming, preliminary wind tunnel tests were performed at Re = 500,000. Surface transition visualisation and pressure measurements show that zigzag type trips of a given size and location are effective at promoting transition, thus preventing the formation of laminar separation bubbles and increasing the effective Reynolds number from the critical regime to the supercritical regime. Extension of these experiments to include three additional tripping methods (wires, roughness strips, CADCUT dots) in a range of sizes, at Reynolds number of 200,000 and below, have been performed in a recirculating water channel. Analysis of surface pressure measurements and time resolved PIV for each trip device, size and location has established a set of recommendations for successful use of tripping for future, low Reynolds number landing gear testing.

1:31PM R4.00003 Discrete surface roughness effects on a blunt hypersonic cone in a quiet tunnel , NICOLE SHARP, EDWARD WHITE, Texas A&M University — The mechanisms by which surface roughness creates boundary-layer disturbances in hypersonic flow are little understood. Work by Reshotko (AIAA 2008-4294) and others suggests that transient growth, resulting from the superposition of decaying non-orthogonal modes, may be responsible. The present study examines transient growth experimentally using a smooth 5-degree half-angle conic frustum paired with blunted nosetips with and without an azimuthal array of discrete roughness elements. A combination of hotwire anemometry and Pitot measurements in the low-disturbance Mach 6 Quiet Tunnel are used for boundary layer profiles downstream of the ring of roughness elements as well as azimuthal measurements to examine the high- and low-speed streaks characteristic of transient growth of stationary roughness-induced disturbances.

1:44PM R4.00004 ABSTRACT WITHDRAWN –

1:57PM R4.00005 ABSTRACT WITHDRAWN –

2:10PM R4.00006 Linear and nonlinear receptivity of the boundary layer in transonic flows , ANATOLY RUBAN, MARINA KRAVTSOVA, TOMASS BERNOTS, Imperial College London — This paper is concerned with the phenomenon of the generation of Tollmien-Schlichting waves in laminar boundary layer on an aircraft wing in transonic flow regime. A particular form of the boundary layer receptivity is considered when the boundary layer encounters a local roughness on the wing surface in the form of a gap, step or a hump. We assume that the boundary layer is exposed to acoustic noise and study the interaction of the acoustic waves with the flow perturbations produced in the boundary layer by the roughness. Two approaches are used. The first one is theoretical; it is based on large Reynolds number asymptotic analysis of the Navier-Stokes equations leading to the transonic version of the triple-deck theory. Under assumptions that the acoustic noise level is weak, and the roughness height is small, and analytic formula for the amplitude of the generated Tollmien-Schlichting wave is deduced. In the second, numerical, approach the restriction on the roughness height is lifted, which allows for the flows with local separation near the roughness to be considered. The calculations have been perform for different value of the Karman number, and we found that in the flow separation always leads to a significant enhancement of the receptivity process.

2:23PM R4.00007 Numerical investigation of boundary layer receptivity to free-stream disturbances and surface excrescences , ADRIAN SESCU, Mississippi State University, MIGUEL VISBAL, DONALD RIZZETTA, Air Force Research Laboratory — In this study, the receptivity of boundary layers to surface imperfections and free-stream disturbances is analyzed in two-dimensions using a high-fidelity Navier-Stokes solver based on high-order compact spatial schemes, and implicit time integration. The surface imperfection is an idealized form of typical excrescences that exist on the surface of aircraft wings, while the free-stream disturbances mimic real perturbations existing in the atmosphere or in wind tunnels. The geometry consists of an superelliptic-leading-edge flat plate with a forward or rearward excrescence on its surface. Acoustic and vortical waves are generated using a source term, as opposed to using inflow boundary conditions, to avoid spurious waves that may propagate from boundaries. The results show that the acoustic waves are very efficient in exciting the Tollmien-Schlichting (TS) waves downstream of the step, as expected, and that the wavelength of TS waves scales linearly with the wavelength of the acoustic waves. The vorticity waves are less likely to excite the TS waves, but when they do so the TS waves are grouped in wave packets that are consistent with the wavelength of the vorticity waves. Other relevant results will be included and discussed.

2:36PM R4.00008 Effect of porous surface on pre-transitional supersonic boundary-layer disturbances generated by free-stream vortices , PIERRE RICCO, The University of Sheffield — A supersonic laminar flat-plate boundary layer at Mach number M=6 flowing over a porous surface is studied numerically and by asymptotic methods. The flow is perturbed by small-amplitude free-stream vortical disturbances of the convective gust type. These external agents generate streamwise-elongated low-frequency disturbances of the kinematic kind, i.e. compressible streaky Klebanoff modes, and of the thermal kind, i.e. thermal streaks, which grow algebraically downstream. For boundary layer fluctuations with a spanwise wavelength comparable with the boundary layer thickness, the porous surface has a negligible effect on the growth and evolution of the streaks. When the spanwise wavelength is instead larger than the boundary layer thickness, the disturbances are effectively attenuated by the porous surface. For a specified set of frequency and wavelengths, the streaky structures evolve into oblique Tollmien-Schlichting waves through a leading-edge-adjustment receptivity mechanism. The growth rate of these waves increases slightly over the porous set, thus confirming previous results obtained through stability analysis. Our receptivity analysis allows us to calculate the wave amplitude, which is attenuated by the porous surface. Further asymptotic analysis based on triple-deck theory confirms the numerical results.

2:49PM R4.00009 Experimental Investigation on the Effects of Free-Stream Turbulence on Swept-Wing Transition , TARIQ SAEED, JONATHAN MORRISON, Imperial College London — This study focusses on the experimental investigation of the high-frequency secondary instability development of roughness-induced stationary crossflows on a swept wing in moderate levels of free-stream turbulence. The first phase of this study is to determine the magnitude of the vortices produced for comparison with the predictions. The evolution of the disturbances are measured as they evolve downstream. The "AERAST" model used in this study is designed to enhance the growth of the crossflow instability, and has a sweep angle of $40°$. The model has been tested in the Department's 10' x 5' wind tunnel which has a top speed of 40 m/s, corresponding to $Re_c = 2.2 \times 10^6$. The measured streamwise and lateral free-stream turbulence levels of the facility are 0.13% and 0.21%, respectively; an additional turbulence reduction screen to be installed would give 0.05% and 0.12%, respectively. A sublimation flow-visualisation technique is used to assess transition location, and detailed hot-wire measurements are conducted to assess disturbance evolution.

3:02PM R4.00010 Interpretations of Incompressible Continuous Spectrum Receptivity Curves for Transient Growth[1] , JASON MONSCHKE, EDWARD WHITE, Texas A&M University — Receptivity of transient disturbances to distributed surface roughness is not representable as a single value but is instead a complex-valued function with a different value for each continuous spectrum mode of the Orr–Sommerfeld/Squire equations. Specific characteristics of the curves give rise to streamwise vorticity of varying strength and at different locations within the boundary layer. The various combinations of streamwise vorticity and the initial streamwise velocity disturbance result in the many types of energy evolution seen in experiments and DNS. Following the work of Tumin [Phys. Fluids 15, 2525 (2003)], Denissen & White [Phys. Fluids 21, 114105 (2009)] developed a technique to decompose experimental measurements made downstream of a roughness array into the constituent continuous spectrum modes. These techniques provide for significant data reduction because receptivity curves encode the complete downstream evolution of the laminar boundary layer. Even though the decomposition uniquely characterizes receptivity to roughness, the small set of measured curves has hindered a systematic understanding of the physical meaning of roughness receptivity curves. Our findings help to associate specific receptivity-curve shapes with physically observable behavior.

[1]Supported by AFOSR under grant FA9550-11-1-0203

Tuesday, November 26, 2013 1:05PM - 3:15PM –

Session R5 CFD VIII: Numerical Methods II 327 - Oleg V. Vasilyev, University of Colorado

1:05PM R5.00001 High-order discontinuous-Galerkin simulations of flows over airfoils with curved boundaries[1] , DANIEL NELSON, GUSTAAF JACOBS, San Diego State University — We compare the flow around a NACA 65-(1)412 airfoil using straight sided and curved sided boundary subdomains in a discontinuous-Galerkin spectral element computation. Specifically, we examine the structure of the vortex street wake and note significant differences in the wake dynamics between the two boundary subdomain implementations. At a Reynolds number of 20,000, the boundary layer on the suction side of the airfoil separates at approximately 60% of the cord length behind the leading edge. The resulting unstable shear layer interacts with vortices generated at the trailing edge to form a vortex street wake. When the subdomain boundary is fitted to the airfoil spline with a curved side, the location of the separation point is fixed and the vortex street is regular and periodic. When straight-sided subdomains are used, the separation point alternates erratically between subdomain corners, resulting in an aperiodic roll-up of the shear layer and subsequent aperiodicity in the near and far wake.

[1]This work was supported in part by Solar Turbines

1:18PM R5.00002 Extended Discontinuous Galerkin Methods for tow-phase flows[1] , FLORIAN KUMMER, Center for Turbulence Resarch, Stanford — Multiphase flows of two (or more) immiscible fluids occur in multiple technical relevant application, e.g. pre-mixed combustion, evaporation or sprays. Since density and viscosity are discontinuous at the interface between the two fluids, the pressure and velocity field contain singularities. For material interfaces, the presence of surface tension will induce a jump in the pressure field. Solutions for non-material interfaces contain jumps in velocity and pressure field, even without any surface tension models. The numerical treatment of these jumps, and the high gradients which are induced by them, is challenging. This is especially true for high-order methods, like the discontinuous Galerkin (DG) method, that usually show their major advantage–the high convergence order–only for smooth solutions. We will present an Extended Discontinuous Galerkin (XDG) method that is able to represent jumps mentioned above with sub-cell accuracy. The interface between the two fluids is represented by a high-order DG-based level set method. In cells which are cut by the interface, separate degrees-of-freedom are employed for both faces. At the interface, the jump conditions are discretized in a weak sense, in order to couple both phases.

[1]Funded by the German Research Association (DFG).

1:31PM R5.00003 Eliminating resonances in the Galerkin-truncated Burgers and Euler equations using wavelet filtering , KAI SCHNEIDER, M2P2-CNRS & Aix Marseille University, 38 rue F. Joliot-Curie, 13451 Marseille Cedex 20, France, RODRIGO PEREIRA, Inmetro & Instituto de Fisica, Universidade Federal do Rio de Janeiro, CP 68528, 21945-970, Rio de Janeiro, RJ, Brazil, ROMAIN NGUYEN VAN YEN, Fachbereich Mathematik und Informatik, Freie Universitaet Berlin, Arnimallee 6, D-14195 Berlin, Germany, MARIE FARGE, LMD-CNRS-IPSL, Ecole Normale Superieure, 24 rue Lhomond, 75231 Paris Cedex 5, France — It is well known that solutions to the Fourier-Galerkin truncation of the inviscid Burgers equation (and other hyperbolic conservation laws) do not converge to the physically relevant entropy solution after the formation of the first shock. This loss of convergence was recently studied in detail in [S.S. Ray et al., Phys. Rev. E 84, 016301 (2011)], and traced back to the appearance of a spatially localized resonance phenomenon perturbing the solution. In this work, we propose a way to cure this resonance by filtering a wavelet representation of the Galerkin-truncated equations. A method previously developed with a complex-valued wavelet frame is applied and expanded to embrace the use of real-valued orthogonal wavelet basis, which we show to yield satisfactory results only under the condition of adding a safety zone in wavelet space. We also apply the complex-valued wavelet based method to the 2D Euler equation problem, showing that it is able to filter the resonances in this case as well. For details we refer to R. Pereira et al., Phys. Rev. E 87, 033017, 2013.

1:44PM R5.00004 Model Order Reduction for the Coupled System of Flow and Moving Structure[1] , HAOTIAN GAO, MINGJUN WEI, New Mexico State University — Traditional POD-Galerkin projection, as a popular approach for model reduction, is applied in a fixed fluid domain which, however, is not the case for many fluid-solid systems with moving objects/boundaries. Instead of treating a time-dependent fluid domain, we consider the combination of fluid and solid one single stationary domain. The idea, which is similar to immersed boundary technique used in numerical simulation, is having original Navier-Stokes equation in the combined and fixed fluid-solid domain and adding extra body-force terms to the equation only in solid area to represent the moving boundary or solid structure. Global POD modes can then be achieved with a special inner product also defined in the combined domain. With both the modes and equations defined in a fixed fluid-solid domain, the Galerkin projection is applied directly in the same domain and provides a global reduced-order model for the system. In comparison to the traditional approach, the new global model ends up with extra terms to represent solid motion and our preliminary results have shown that these terms are critical in sustaining system energy.

[1]Supported by ARL

1:57PM R5.00005 A Robust Integration Method for Stiff Transport Equations , JOSE ESCOBAR, ISMAIL CELIK, West Virginia University — Numerical simulation of reactive flows is one of the most difficult problems in Computational Fluid Dynamics (CFD). The difficulties are mainly due to the wide range of characteristic time scales present in the mass production/consumption sources which lead to a stiff system of governing equations. Classical integration methods such as explicit Euler method are restricted by the smallest characteristic time scale, and the explicit Runge-Kutta (RK) methods require intermediate predictor-corrector steps which make the problem computationally expensive. Implicit methods are also computationally expensive due the calculation of the Jacobian which makes their implementation difficult for reactive systems with tens of chemical species and hundreds of reactions. The current study explores the possibility of solving the transport equations for species concentrations faster than the standard methods without compromising accuracy. The present approach is based on transformation of the variable of interest using the hyperbolic tangent function. The proposed transformation also has the advantage of ensuring that the value of the variable of interest to be always in the range from zero to one, which is highly desirable when solving for specie mass or molar fractions. The possibility of using much larger time steps compared to the classical methods is an additional advantage.

2:10PM R5.00006 Level Set Jet Schemes for Stiff Advection Equations , DAVID SALAC, EBRAHIM KOLAHDOUZ, University at Buffalo SUNY — The stable and accurate modeling of stiff multiphase fluid systems represent a major challenge. In this talk strategies to employ the Jet Level Set scheme of Nave et. al. in stiff advection problems, such as lipid bilayer vesicle simulations, are presented. The rational and sample implementations of these methods will be shown. The results will demonstrate that the strict time-step restrictions can be alleviated without greatly reducing the accuracy of the method.

2:23PM R5.00007 An Improved Advection Scheme for Implicit Interfaces on Cartesian Grids , ZHIPENG QIN, AMIR RIAZ, University of Maryland, ELIAS BALARAS, The George Washington University, KEEGAN DELANEY, Virginia Tech — A new approach is presented to reinitialize the level set function by a direct projection of interface topology. The interface topology is built with the help of 2nd order interpolation reconstruction and interface smoothing by least squares fitting scheme. With the help of three kinds of errors the new approach is shown to both conserve mass and maintain topology during advection. Existing methods for the advection of implicit interfaces, such as classical redistance techniques and interface recompression methods, are compared with the new scheme. The improvement achived by the new scheme is demonstrated on uniform Cartesian grids for 2D vortex and two-phase incompressible rising bubble cases.

2:36PM R5.00008 An Iterative Brinkman penalization for particle vortex methods , J.H. WALTHER, M.M. HEJLESEN, Technical University of Denmark, A. LEONARD, California Institute of Technology, P. KOUMOUTSAKOS, ETH Zurich — We present an iterative Brinkman penalization method for the enforcement of the no-slip boundary condition in vortex particle methods. This is achieved by implementing a penalization of the velocity field using iteration of the penalized vorticity. We show that using the conventional Brinkman penalization method can result in an insufficient enforcement of solid boundaries. The specific problems of the conventional penalization method is discussed and three examples are presented by which the method in its current form has shown to be insufficient to consistently enforce the no-slip boundary condition. These are: the impulsively started flow past a cylinder, the impulsively started flow normal to a flat plate, and the uniformly accelerated flow normal to a flat plate. The iterative penalization algorithm is shown to give significantly improved results compared to the conventional penalization method for each of the presented flow cases.

2:49PM R5.00009 Characteristic-based Volume Penalization Method for Arbitrary Mach Flows Around Solid Obstacles[1] , NURLYBEK KASIMOV, ERIC BROWN-DYMKOSKI, OLEG VASILYEV, University of Colorado Boulder — A new volume penalization method to enforce boundary conditions on solid/moving/deformable obstacles of arbitrary shape for both Navier-Stokes and Euler equations is presented. The approach extends the Brinkman penalization to generalized Neumann and Robin boundary conditions by introducing hyperbolic penalization terms with characteristics pointing inward on solid obstacles. The boundary conditions for both integrated and non-integrated variables can be imposed in a systematic manner that parallels the prescription of exact boundary conditions. A principle advantage of the method is that it provides a systematic means of controlling the error. This approach is general and applicable to a wide variety of flow regimes. Examples of supersonic/subsonic viscous/inviscid complex geometry flows are given and discussed.

[1]This work was supported by ONR MURI on Soil Blast Modeling.

3:02PM R5.00010 A conservative adaptive wavelet method for the shallow water equations on the sphere , NICHOLAS KEVLAHAN, MATTHIAS AECHTNER, McMaster University, THOMAS DUBOS, Ecole Polytechnique — This talk presents the first dynamically adaptive wavelet method for the shallow water equations on a staggered hexagonal C-grid on the sphere. Pressure is located at the centres of the primal grid (hexagons) and velocity is located at the edges of the dual grid (triangles). Distinct biorthogonal second generation wavelet transforms are developed for the pressure and the velocity. These wavelet transforms are based on second-order accurate interpolation and restriction operators. Together with compatible restriction operators for the mass flux, circulation and Bernoulli function, they ensure that mass is conserved and that there is no numerical generation of vorticity when solving the shallow water equations. The shallow water equations are discretized on the dynamically adapted multiscale grid using a mass and potential-enstrophy conserving finite-difference scheme. The method is applied to a zonal jet test case, turbulence on the rotating sphere and western boundary current flow. Solid boundary conditions are implemented using a new multiscale penalization of the shallow water equations.

Tuesday, November 26, 2013 1:05PM - 3:02PM –

Session R6 Nanofluids III 328 - John Shelton, Carnegie Mellon University

1:05PM R6.00001 Towards realistic multiscale molecular-continuum modelling of water flow through nanotube membranes , KONSTANTINOS RITOS, MATTHEW BORG, Mechanical & Aerospace Engineering, University of Strathclyde, Glasgow G1 1XJ, UK, DUNCAN LOCKERBY, School of Engineering, University of Warwick, Coventry CV4 7AL, UK, SPELA IVEKOVIC, YONGHAO ZHANG, JASON REESE, Mechanical & Aerospace Engineering, University of Strathclyde, Glasgow G1 1XJ, UK — We present a new hybrid molecular-continuum method for modelling the nano flows inside micrometer-thick membranes. Our aim is to obtain results for practical filtration membranes that are as accurate as molecular dynamics (MD) and at the same time significantly more computationally efficient. Computational savings are obtained by replacing long nanotube sections - that are highly scale-separated - by much smaller but representative MD simulations, without any substantial loss of accuracy. These individual MD simulations are coupled together via standard continuum fluid-dynamics equations that dictate the overall macroscopic flow in the membrane. For this specific problem we use the conservative continuity and momentum equations as we consider the flow isothermal, incompressible and low-speed. Our iterative algorithm computes at each iteration the new constraints on the pressure differences applied to individual micro elements, and enforces overall continuity within the membrane. Validation tests are through direct comparison with full MD simulations of 50 and 150 nm thick membranes. We show results for micrometer-thick membranes and compare our predictions with previously-published experimental data.

1:18PM R6.00002 Molecular dynamics study of instability of nano-liquid column , TAKERU YANO, TAICHI MURAKAMI, Department of Mechanical Engineering, Osaka University — Molecular dynamics study of argon nano-liquid column in equilibrium with surrounding its vapor is executed to investigate the instability for axisymmetric disturbances. As a result of careful and accurate calculation for nano-liquid columns of radius 1 to 4 nm, we can demonstrate that the classical criterion of stability limit $L_z = 2\pi R$ holds for the nano-columns with the appropriate definition of the column radius R for columns of length L_z.

1:31PM R6.00003 Study of contact line motion in two phase flow using molecular dynamics[1] , JOSEPH THALAKKOTTOR, KAMRAN MOHSENI, University of Florida — Contact line motion is an age old problem. The scale in which the continuum assumption breaks down resulting in the motion of contact line makes it difficult to analyze the problem. Using molecular dynamic simulations we intend to investigate the cause and effects of contact line motion. The results indicates that miscibility between the two fluids, hydrophobicity between fluid and wall, and shear rate of the fluid are some of the key parameters that determine the amount of slip at the triple contact point. Circulation inside a droplet is observed even at nanoscales and is seen to vary inversely with slip length. We also observe non-Newtonian behavior of fluid in the vicinity of the triple contact point. Understanding the affects of these phenomena on contact line motion would help in better understanding the movement of triple contact point in two phase flow. This information would aide in developing a slip model for the triple contact point.

[1]This research was supported by Office of Naval Research

1:44PM R6.00004 CO_2 separation using a porous graphene/IL membrane , JOONHO LEE, NARAYANA R. ALURU, Department of Mechanical Science and Engineering, University of Illinois at Urbana-Champaign, Urbana, IL 61801 — We investigated the separation of CO_2 from CO_2/O_2 mixture using a porous graphene/ionic liquid (IL) membrane. By performing extensive non-equilibrium molecular dynamics (NEMD) simulations, we observed dramatic CO_2 separation with a 0.99 nm graphene nanopore. By calculating the density distribution, we show that a strong layering of the ionic liquid is observed near the 0.99 nm graphene nanopore, while such a strong layering is not observed in the larger 2.22 nm diameter pore. The strong layering induces a near perfect blockage of O_2 molecules from diffusing into the ionic liquid. Void analysis shows that the layering of ionic liquid serving as a blockage for O_2 molecules does not hinder CO_2 solvation in the ionic liquid.

1:57PM R6.00005 Effect of air on water capillary flow in silica nanochannels , HARVEY ZAMBRANO, Universidad de Concepcion, JENS WALTHER, Technical University of Denmark, ELTON OYARZUA, Universidad de Concepcion — Capillarity is a classical topic in fluid dynamics. The fundamental relationship between capillarity and surface tension is solidly established. Nevertheless, capillarity is an active research area especially as the miniaturization of devices is reaching the molecular scale. Currently, with the fabrication of microsystems integrated by nanochannels, a thorough understanding of the transport of fluids in nanoconfinement is required for a successful operation of the functional parts of such devices. In this work, Molecular Dynamics simulations are conducted to study the spontaneous imbibition of water in sub 10 nm silica channels. The capillary filling speed is computed in channels subjected to different air pressures. In order to describe the interactions between the species, an effective force field is developed, which is calibrated by reproducing the water contact angle. The results show that the capillary filling speed qualitatively follows the classical Washburn model, however, quantitatively it is lower than expected. Furthermore, it is observed that the deviations increase as air pressure is higher. We attribute the deviations to amounts of air trapped at the silica-water interface which leads to changes in the dynamics contact angle of the water meniscus.

2:10PM R6.00006 Dissimilar viscosity induced sample pre-concentration in elecrokinetic nanofluidic channels , DEAN WINK, ELIJAH SHELTON, SUMITA PENNATHUR, University of California Santa Barbara, BRIAN STOREY, Franklin W. Olin College of Engineering — Nanofluidic analysis systems boast many advantages: portability, small sample handling, short processing times, and potential for integration with mobile electronics. However, such systems face the challenge of detecting increasingly small volumes of sample at low concentrations. In this work, we demonstrate a unique pre-concentration technique in electrokinetic nanofluidic systems based on a viscosity mismatch between two fluids. In nanofluidic electrokinetic systems, finite electric double layers (EDL) lead to non-uniform electric potentials and transverse concentration distributions. Therefore, when the EDL is comparable in size to the channel height, negatively charged ions are repelled from negatively charged walls and preferentially populate the channel centerline. Furthermore, an axial piecewise viscosity distribution induces internal pressure gradients within the channel. These force the ions to move at a different average velocities based on the pressure gradient being favorable or adverse, leading to focusing. To experimentally probe this phenomenon, we electrokinetically inject solutions of borate buffer with and without glycerol (to change the viscosity) and use a fluorescent tracer dye to visualize the flow. We perform the injections in cross-geometry channels of 20 micron, 1 micron, and 250 nanometer depths. We measure fluorescence at 5, 10 and 15 mm distances from junction. Enhancement is characterized by comparing intensities to control measurements for systems with uniform viscosity.

2:23PM R6.00007 Overlimiting current through ion concentration polarization layer: Experimental verifications of the surface conduction mechanism[1] , SUNGMIN NAM, Seoul National University, JOONSEONG HEO, GEUNBAE LIM, Pohang University of Science and Technology, SUNG JAE KIM, Seoul National University — The mechanism of overlimiting current which observed in most practical electrochemical membrane systems has become a critical issue that numerous researchers have tried to resolve. Overlimiting current is closely related to the ion concentration polarization phenomenon which represents an imbalance of ion concentrations nearby the membrane. Among a number of studies, a recent theoretical study suggested that surface conduction is the core mechanism, while an electro-convective flow and water dissociation has been regarded as the major mechanism of overlimiting current. In this presentation, we provide rigorous experimental evidences of the role of surface conduction using a micro/nanofluidic platform. Conclusively the surface conduction enhances the overlimiting current characteristics, while the electro-convection retards the initiation of overlimiting current. Thus, this result can contribute to the advances for both understanding a fundamental electrokinetic theory and engineering applications based on the ion concentration polarization phenomena.

[1]This work is supported by Basic Science Research Program (2013R1A1A1008125) and Future-based Technology Development Program (Nano Fields) (2009-0082952) by MISP and by MOE (No. 2012R1A2A2A06047424), Korea.

2:36PM R6.00008 A Landau-Squire Nanojet[1] , SANDIP GHOSAL, Dept. of Mech. Eng. & (by courtesy) Eng. Sci. Appl. Math., Northwestern University, USA, NADANAI LAOHAKUNAKORN, Dept. of Physics, Cambridge University, UK, BENJAMIN GOLLNICK, FERNANDO MORENO-HERRERO, Centro Nacional de Biotecnologia, Spain, DIRK G.A.L. AARTS, ROEL P.A. DULLENS, Dept. of Chemistry, Oxford University, UK, ULRICH F. KEYSER, Dept. of Physics, Cambridge University, UK — Fluid jets are found in nature at all length scales – microscopic to cosmological. Here we report on what may be the smallest liquid jet ever observed: an electroosmotically driven flow from a single glass nanopore about 75 nm in radius with a maximum flow rate of about 30 pL/s. A novel anemometry technique allows us to map out the vorticity and velocity fields which show excellent agreement with the classical Landau-Squire solution of the Navier Stokes equations for a point jet. We observe a phenomenon that we call flow rectification: an asymmetry in the flow rate with respect to voltage reversal. Such a nanojet could potentially find applications in gene delivery, nano patterning, and as a diode in microfluidic circuits.

[1]NIH (R01HG004842) & Leverhulme Trust (SG), George & Lillian Schiff Foundation & Trinity College (NL), Ministry of Education, Spain (BG), Deutsche Froschungsgemeinschaft & European Research Council (UFK)

2:49PM R6.00009 ABSTRACT WITHDRAWN –

Tuesday, November 26, 2013 1:05PM - 2:49PM –

Session R7 Multiphase Flows IX 329 - Casey Harwood, University of Michigan

1:05PM R7.00001 Experimental Investigation of Ventilation of a Surface Piercing Hydrofoil[1] , CASEY HARWOOD, FRANCISCO MIGUEL MONTERO, YIN LU YOUNG, STEVEN CECCIO, University of Michigan — Bodies that pierce a liquid free-surface are prone to entrainment of atmospheric and/or vaporous gases. This process, called ventilation, can occur suddenly and violently, drastically altering hydrodynamic response. Experiments have been conducted at the free-surface towing-tank in the University of Michigan Marine Hydrodynamics Laboratory to investigate fully attached, partially ventilated, and fully ventilated flows around a canonical surface-piercing hydrofoil. The objectives of the work are: (i) to gain a broad and improved understanding of the physics of ventilation, (ii) to classify the physical mechanisms by which ventilation inception and washout may occur and quantify the conditions required for each mechanism and (iii) to quantify the effects of ventilation on global hydrodynamic responses, including the six force and moment components. Experimental data and high-speed video will be used to illustrate the impact of ventilation on hydrodynamic loads, pressures, and flow structures. The completion of this study is expected to contribute significantly toward a comprehensive understanding of ventilation physics, and toward an improved ability to design safe and controllable ventilated lifting surfaces for use in propulsion, energy harvesting, and turbomachinery.

[1]Supported by: The Office of Naval Research (ONR) (Grant no. N00014-09-1-1204); the National Research Foundation of Korea (NRF) (GCRC-SOP Grant no. 2012-0004783); the National Science Foundation Graduate Student Research Fellowship (Grant No. DGE 1256260)

1:18PM R7.00002 Influence of scaling effects in the ventilation of surface-piercing bodies[1] , FRANCISCO MIGUEL MONTERO, CASEY M. HARWOOD, YIN LU YOUNG, STEVEN L. CECCIO, Dpt. of Naval Architecture and Marine Engineering - U. of Michigan — Ventilation is a process by which atmospheric air is entrained to the submerged portion of a body. The study of this process is of critical importance to the design and control of surface piercing bodies, such as hydrofoils, propellers, struts and turbines, as it can result in a very sudden variation of the forces acting on the body. Ventilation is also influenced by the presence of vaporous cavitation. This cavitation-ventilation process can be complex, and in order to replicate it experimentally, there are several scaling issues that must be carefully considered. The objectives of this work are: (1) to characterize the different ventilation mechanisms, the various parameters that influence the ventilation process, as well as the resultant impact on performance, (2) to discuss and quantify scaling effects in model tests in towing tank and cavitation tunnel studies, and (3) to discuss additional research needs in terms of experimental and numerical modeling. Theoretical arguments, as well as prior and new experimental data, will be used to present the different ventilation mechanisms and derive and illustrate the related scaling issues in both cavitation tunnel and towing tank studies.

[1]Research funded by ONR Grant No. N00014-11-1-0849 & the National Research Foundation of Korea (NRF) (GCRC-SOP Grant no. 2012-0004783).

1:31PM R7.00003 Forced drainage in a 2D foam in a microfluidic system using thermocapillary stress , MARIE-CAROLINE JULLIEN, VINCENT MIRALLES, Gulliver, CNRS, ESPCI Paris-Tech, 10 rue Vauquelin, F-75005 Paris, France, BERTRAND SELVA, LOF, unité mixte Rhodia–CNRS–Bordeaux 1, 178 avenue du Docteur Schweitzer, F–33608 Pessac cedex, France., JULIEN MARCHALOT, Institut des Nanotechnologies de Lyon, INL, CNRS UMR5270, France, ISABELLE CANTAT, IPR, (UMR CNRS 6251), Université de Rennes 1, 35000 Rennes, France, MMN-ESPCI/CNRS TEAM, LOF COLLABORATION, INL COLLABORATION, IPR COLLABORATION — We present an experimental configuration allowing the possibility to control the liquid fraction in a 2D microfoam located in a Hele-Shaw cell. A Marangoni stress at the air-water interface is generated by applying a constant temperature gradient in situ, and leads to the drainage of the liquid phase. First, in order to avoid gravity drainage, the cell is placed horizontally and we are able to drain up to 70% of the liquid phase, for foams of initial liquid fraction $\phi_0 \sim 15\%$. Next, the cell is placed vertically and the Marangoni stress for temperature gradients above 3.1K.mm^{-1} is strong enough to counterbalance gravity drainage. Finally, a mass conservation approach based on scaling arguments and numerical simulations giving access to the velocity profile in a pseudo-Plateau border happen to be in very good agreement with the experimental results, showing that we can accurately control the liquid fraction in a 2D microfoam.

1:44PM R7.00004 COST Action FP1005 "Fibre suspension flow modelling" , CRISTIAN MARCHIOLI, University of Udine — Fibre suspensions are extremely complex solid-liquid systems since their components (fibres, flocs, air bubbles and additives) interact mutually in a complex way. The dynamics of fibre suspensions are crucial in many real-life applications, such as pulp and paper production. Current understanding of suspension flow dynamics remains poor and incomplete, resulting in conservative design of industrial equipments, low energy efficiency and equipment oversizing. In this paper, the most recent advancements in modelling and experimentation of fibre suspensions dynamics are presented. These advancements have been obtained in the framework of Action FP1005, funded by the COST Programme (European Cooperation in Science and Technology) to coordinate nationally-funded research on a European level. The Action aims at developing and validating numerical models for prediction of fibre suspensions as well as measurement techniques. The Action offers a forum to solve test cases and to compare simulated results to experiments, resulting in more reliable simulation tools to industry. Successfull introduction of such tool into industrial practice is crucial to innovate and increase competitivity of papermaking industry.

1:57PM R7.00005 ABSTRACT WITHDRAWN –

2:10PM R7.00006 An ensemble method for targeted adaptive observations applied to multi-phase flows[1] , ZHIZHAO CHE, FANGXIN FANG, JAMES PERCIVAL, GEOFFREY HEWITT, CHRISTOPHER PAIN, OMAR MATAR, Imperial College London, MICHAEL NAVON, Florida State University — Many flow problems, such as turbulence and multiphase, are extremely complex due to their strong nonlinearity. It is important to simulate and measure different parameters accurately, such as pressure drops and flow rates, to which flow phenomena are very sensitive. Therefore, it is essential to put the sensors at the locations with larger impact, and to avoid locations with lower impact. Here, we proposed an ensemble method to estimate the impact of observations at different locations. Ensembles were generated by adding perturbations to the initial conditions, say. Different target functions were used to quantify the impact of observations. In comparing with other methods for estimating impact, this ensemble method is very simple to implement, and is independent of the definition of the target functions. This method is demonstrated by applying it to the one-dimensional Burgers equation. The next steps are to extend this method to various complex problems such as multiphase flows.

[1]EPSRC Programme Grant EP/K003976/1

2:23PM R7.00007 Generation of Pulsating Supercavities in a Rigid Wall Water Tunnel , GRANT SKIDMORE, TIMOTHY BRUNGART, JULES LINDAU, The Pennsylvania State University Applied Research Laboratory — The use of ventilated supercavities for underwater travel has many potential benefits, but before they can be fully exploited, obstacles in safe generation of a supercavitating body must first be overcome. The principle obstacle in this is determining the closure regime of the supercavity (re-entrant jet, twin vortex, or pulsating) from a given set of testing conditions. The re-entrant jet and twin vortex closure regimes are stable and should not create problems for the supercavitating body. Supercavity pulsation, however, is a self-excited resonance phenomenon that destabilizes the supercavity and leads to the periodic release of gas pockets at the tail of the cavity. Thus the phenomenon needs to be fully studied in a controlled environment. However, there are unanswered questions as to whether the pulsatory phenomenon may be properly obtained in a rigid-walled, closed-circuit water tunnel. Utilizing the 0.305 m and 1.219 m diameter water tunnels at ARL Penn State, an experimental study on both the pulsation phenomenon and the effects of tunnel blockage has been conducted. Here, we detail the findings of this study and discuss possible ways to mitigate the pulsation phenomenon.

2:36PM R7.00008 The flow mechanism causing performance breakdown in cavitating axial turbomachines[1] , DAVID TAN, RINALDO MIORINI, Johns Hopkins University, ELENA VAGNONI, Politecnico di Milano, IAN WILKES, JOSEPH KATZ, Johns Hopkins University — Cavitation degrades the performance of pumps, eventually leading to complete performance breakdown. Identifying the mechanisms causing breakdown has been a long-standing challenge. Using high-speed imaging (9.6 kHz) and pressure fluctuation measurements in the JHU optically refractive index matched facility, we elucidate the cavitation breakdown process in a waterjet pump. It involves interactions of a cavitating tip leakage vortex with the trailing edge of attached cavitation on the rotor blade, specifically, entrainment of the vortical cloud cavitation by the tip vortex. As the pressure is decreased, the blade suction side (SS) sheet cavitation extends further downstream. When the trailing edge extends to the region where the rotor blades overlap, the entrained sheet cavitation vorticity forms a vortex with axis aligned perpendicularly to the blade, and extending over the entire passage. Decreasing the pressure increases the size of this vortex and generates several parallel structures. The process causes a sharp drop in the pressure difference across the blade tip, i.e. the blade loading (performance) diminishes. Examination of previously published images of cavitation in rocket inducers suggests that this phenomenon occurs in other axial turbomachines as well.

[1]Sponsored by ONR

Tuesday, November 26, 2013 1:05PM - 2:10PM –

Session R8 General Fluid Dynamics IV: Theory II 330 - John Dannenhoffer, Syracuse University

1:05PM R8.00001 Data Fusion for Fluid Dynamic Data , CHRISTOPHER RUSCHER, JOHN DANNENHOFFER, MARK GLAUSER, Syracuse University — In recent years, fluid dynamic measurement tools and computational fluid dynamics (CFD) have greatly improved, leading to vast amounts of data being collected. Extracting the useful information from large data sets can be a challenging task when investigating data from a single source. However, most experiments use data from multiple sources such as particle image velocimetry (PIV), pressure sensors, acoustic measurements, and CFD to name a few. Knowing the strengths and weaknesses of each measurement technique, one can fuse the data together to improve the understanding of the problem being studied. Using data fusion, sensor fusion, and data integration techniques in combination with fluid dynamic analysis tools, one can extract information from large multi-source fluid dynamic data sets, which is not obtainable from any single data source alone.

1:18PM R8.00002 Dynamics and Control of the 2-d Navier-Stokes Equations[1] , NEJIB SMAOUI, MOHAMED ZRIBI, Kuwait University — The control problem of the dynamics of the two-dimensional (2-d) Navier-Stokes (N-S) equations with spatially periodic and temporally steady forcing is studied. First, we devise a dynamical system of several nonlinear differential equations by a truncation of the 2-d N-S equations. Then, we study the dynamics of the obtained Galerkin system by analyzing the system's attractors for different values of the Reynolds number, R_e. By applying the symmetry of the equation on one of the system's attractors, a symmetric limit trajectory that is part of the dynamics is obtained. Next, a control strategy to drive the dynamics from one attractor to another attractor for a given R_e is designed. Finally, numerical simulations are undertaken to validate the theoretical developments.

[1]This work was supported and funded by Kuwait University Research Grant No. SM02/13.

1:31PM R8.00003 K-Means Clustering for Data Visualization and Flow Interpretation: Inclined Jet in Crossflow Example , JULIA LING, JULIEN BODART, FILIPPO COLETTI, JOHN EATON, Stanford University — The k-means clustering algorithm is a versatile data processing technique that has not yet been extensively applied to fluids data sets. The clustering algorithm can operate in high dimensional spaces to extract structure from large data sets. In the context of fluid dynamics, k-means clustering can be used to cluster the output of experimental or computational results based on mean velocity gradients or other single-point statistics. This technique has been applied to three dimensional mean velocity fields for an inclined jet in crossflow that were acquired using MRI-based 3D velocimetry, a Reynolds Averaged Navier Stokes (RANS) simulation, and a Large Eddy Simulation (LES). In each case, the clusters were based on the mean velocity gradient tensor. The optimal number of clusters was determined using an external validation technique in which a linear regression was performed within each LES cluster to predict the Reynolds stresses based on the mean velocity gradient. These linear regressions were subsequently evaluated on a validation subset of the LES data, and it was shown that eight clusters gave the lowest validation error. These eight clusters were used to explore the differences in flow structure between experiment, LES, and RANS and to determine which characteristics were associated with higher error in the RANS simulation. It was shown that the RANS Reynolds stresses were least accurate in regions of high strain or high streamwise vorticity.

1:44PM R8.00004 Covariant Formulation of Fluid Dynamics and Estakhr's Material Geodesic Equation, far down the Rabbit hole , AHMAD REZA ESTAKHR, Researcher — "When i meet God, I am going to ask him two questions, why relativity and why turbulence. A. Einstein" You probably will not need to ask these questions of God, I've already answered both of them. $U^\mu = \gamma(c, u(\vec{r}, t))$ denotes four-velocity field. $J^\mu = \rho U^\mu$ denotes four-current mass density. Estakhr's Material-Geodesic equation is developed analogy of Navier Stokes equation and Einstein Geodesic equation. $\frac{DJ^\mu}{D\tau} = \frac{dJ^\mu}{D\tau} + \Gamma^\mu_{\alpha\beta} J^\alpha U^\beta = J_\nu \Omega^{\mu\nu} + \partial_\nu T^{\mu\nu} + \Gamma^\mu_{\alpha\beta} J^\alpha U^\beta$ Covariant formulation of fluid dynamics, describe the motion of fluid substances. The local existence and uniqueness theorem for geodesics states that geodesics on a smooth manifold with an affine connection exist, and are unique. EMG equation is also applicable in different branches of physics, it all depend on what you mean by 4-current density, if you mean 4-current electron number density then it is plasma physics, if you mean 4-current electron charge density then it is $\frac{DJ^\mu}{D\tau} = J_\nu F^{\mu\nu} + \partial_\nu T^{\mu\nu} + \Gamma^\mu_{\alpha\beta} J^\alpha U^\beta$ electromagnetism.

1:57PM R8.00005 Spreading and atomization dynamics of ultrasonically excited droplets , RANGANATHAN KUMAR, University of Central Florida, DEEPU P, SAPTARSHI BASU, Indian Institute of Science — The dynamics of a sessile droplet under the combined influence of standing pressure wave and a constant substrate acceleration is investigated experimentally. The asymmetric acoustic force field results in radial spreading of the droplet. The spreading rate varies inversely with viscosity which is explained using an analytical model. In low viscosity droplets, towards the end of droplet spreading capillary waves grow to form ligaments of varying length and time scales, ultimately leading to droplet disintegration. Proper Orthogonal Decomposition of high speed images from the droplet spreading phase predicts the likelihood of atomization. The different regimes in the life of surface ligaments are identified. Viscous dissipation plays a crucial role in determining the initial ligament momentum and thus the frequency of ligament breakup. However in the current experimental conditions the growth of a typical ligament is governed by inertial and capillary forces and the influence of viscosity in the ligament growth phase is rather negligible. By including the effect of acoustic pressure, a characteristic timescale is deduced which collapses the ligament growth profiles for different fluids on a straight line.

Tuesday, November 26, 2013 1:05PM - 2:10PM –

Session R9 Bubbles VI: Nanobubbles and Acoustics 333 - Jiansheng Feng, Massachusetts Institute of Technology

1:05PM R9.00001 Generation and acoustic characterization of monodisperse lipid-coated microbubbles , MIGUEL A. PARRALES, JUAN M. FERNANDEZ, MIGUEL PEREZ-SABORID, Dept. Ingenieria Aeroespacial y Mecanica de Fluidos, Universidad de Sevilla, Spain — The acoustic attenuation spectrum for different lipid-coated microbubble suspensions was measured in order to characterize the linear acoustic behavior of ultrasound contrast agents. For that purpose, microbubbles were generated with a very narrow size distribution by using microfluidics techniques operated at two different regimes: flow-focusing and co-flow. We show that monodisperse agents optimize the acoustic echo response by narrowing the attenuation spectrum, which presents a maximum peak for a frequency value corresponding to that of the single bubble resonance. In consequence, the low polydispersity index of our samples increases the accuracy in the estimation of the lipid shell viscoelastic properties. As it has been reported, the non-linear behavior of the coating makes the viscoelastic parameters to change with the equilibrium bubble radius. Our experimental procedure permits the acoustic measurements to be acquired for virtually single-sized suspensions of bubbles, thus reducing the uncertainty in the estimation when using samples with a broad size distribution. The results show the great advantage of dealing with monodisperse populations rather than polydisperse for the acoustic characterization of ultrasound contrast agents.

1:18PM R9.00002 Theory and simulation of linear wave propagation in bubbly liquids accounting for direct bubble-bubble interactions[1] , DANIEL FUSTER, HUGO MARCUS, JEAN-MARC CONOIR, CNRS (UMR 7190), Universite Pierre et Marie Curie, Institut Jean le Rond d'Alembert, France, TIM COLONIUS, Division of Engineering and Applied Science, California Institute of Technology, Pasadena, California 91125, USA, FRANCOIS COULOUVRAT, CNRS (UMR 7190), Universite Pierre et Marie Curie, Institut Jean le Rond d'Alembert, France — The effect of direct bubble-bubble interactions in the response of bubble clusters remains relatively unexplored. In this work we extend the classical linear theory of wave propagation in bubbly liquids [1] to account for the effect of bubble-bubble interactions on the attenuation and the phase speed of acoustic waves. The results are tested against new simulation results obtained by applying the model presented by Fuster & Colonius [2], which is able to capture the effect of direct bubble-bubble interactions. The results are also compared against data previously reported in the literature.

[1] Linear pressure waves in bubbly liquids: Comparison between theory and experiments. K.W. Commander and A. Prosperetti. J. Ac. Soc. Am. 85. 732-746. 1989.
[2] Modeling bubble clusters in compressible liquids. D. Fuster and T. Colonius. J. Fluid Mech. 688. 352-389. 2011.

[1]This work was supported by the ANR DIAMAN program.

1:31PM R9.00003 Realtime Visualization of Surface Nanobubbles Formation , MANISH ARORA, CHON U. CHAN, CLAUS-DIETER OHL, Nanyang Technological University — Nanobubbles nucleate and reach a stable state on the solid-liquid interface after ethanol-water exchange. Understanding the thermodynamic stability beyond their formation requires observation of their dynamical response. Total internal reflection microscopy [1] allows resolving the dynamics of nanobubbles, i.e. their formation, shrinkage, and coalescence. While the lateral resolution is limited by diffraction, their tens of nanometer height profile can be nicely resolved. Here we report on nanobubble dynamics induced by the exchange process in a microfluidic channel on a glass surface.. The water-ethanol mixing results in non-monotonic changes in the refractive index of the medium which can be inferred from TIRF images. Formation of attached nanobubbles at 50 frames per second is observed during the transition from water-to-ethanol as well as ethanol-to-water. Bubbles dissolve within a second in ethanol but persist in water. Though new nanobubbles keep appearing over several seconds, no further change in their size is observed. However, we find occasionally spontaneous merging of neighbouring nanobubbles which we relate to the liquid flow.

[1] Chan and Ohl, Phys. Rev Lett. 109, 174501 (2012)

1:44PM R9.00004 On the Surface Tension of Nanobubbles[1] , HAIM BAU, JOSEPH GROGAN, MICHAEL NORTON, University of Pennsylvania, FRANCES ROSS, IBM T.J. Watson Research Center — Using our custom-made liquid cell, the nanoaquarium [1], we imaged with a transmission electron microscope the formation, growth, and detachment of single nanobubbles, nucleating in a supersaturated aqueous solution [2]. The supersaturation results from electron-induced radiolysis of water. The bubbles are first observed when their radii are about 20nm and detach when their radii are about 200nm. Based on our experimental data, we determined the bubbles' growth rates as functions of time, and found the growth rates to be highly reproducible and nearly independent of time (and bubbles' radii). Comparison of the theoretical predictions for bubble growth rate with our experimental observations suggests that the surface tension of the bubble-liquid interface must depend on the bubble's radius.

[1] Grogan J., and Bau, H., H., 2010, IEEE/ASME Journal of Microelectromechanical Systems 19 (4) 885-894.
[2] http://arxiv.org/abs/1210.3380

[1]The work was supported, in part, by NSF grants 1066573 and 1129722.

1:57PM R9.00005 Observations of Nanobubble Dynamics with Transmission Electron Microscopy , MEERA KANAKAMMA MOHAN, MANISH ARORA, Division of Physics and Applied Physics, School of Physical and Mathematical Sciences, Nanyang Technological University, Singapore 637371, Singapore, UTKUR MIRSAIDOV, Mechanobiology Institute-Singapore, National University of Singapore, 5A Engineering Drive 1, 117411, Singapore, CLAUS-DIETER OHL, Division of Physics and Applied Physics, School of Physical and Mathematical Sciences, Nanyang Technological University, Singapore 637371, Singapore — Recent developments in transmission electron microscopy (TEM) allow the imaging of liquids with high spatial resolution. Here we report on novel studies of water trapped between two monolayers of graphene sheets. The geometry prevents evaporation of the liquid into the low pressure environment of the TEM while providing excellent electron-optical properties for investigations. The graphene sheets are supported by a conventional TEM grid. We report on the nucleation of bubbles, the coalescence between neighbouring bubbles, rupture of thin liquid filaments, and their slow shrinkage. At a dose rate of 100-155 $e^{-}Å^{-2}s^{-1}$ these events are observed conveniently at video frame rate. The correlation with the local electron beam dose rate suggests that the radiolysis induced by the electron beam is the main driving force for most events. In general, we observed bubbles with lateral sizes between 20nm and 100nm and estimated heights between 6nm and 30nm. Likely, the bubbles connect both graphene sheets. In the absence of the electron beam the nanobubbles do not dissolve completely but surprisingly remain stable for even up to one hour. This resembles the stability of surface attached nanobubbles.

Tuesday, November 26, 2013 1:05PM - 3:28PM –

Session R10 General Fluid Dynamics V 334 - Patrick Weidman, University of Colorado

1:05PM R10.00001 Slippery liquid-infused porous surfaces in fully developed pipe flow[1] , HAFEEZ SULAIMON, MARCUS LEE, LEO HELLSTRÖM, BRIAN ROSENBERG, Princeton University, ALEXANDER SMITS, Princeton University, Monash University, MARCUS HULTMARK, Princeton University — Slippery liquid-infused porous surfaces (SLIPS) are created by locking a thin layer of viscous lubricating oil into a porous surface that is textured at the micro/nano scale, with resulting omniphobicity. The oil layer lies between the solid boundary and the surrounding flow, with the potential to create a partial-slip condition at the boundary. SLIPS therefore offers a new approach to achieve drag reduction. Here, SLIPS is applied to fully developed pipe flow for Reynolds numbers ranging from 600 to 1.8×10^5. The pipe flow facility consists of two test sections, an untreated and a SLIPS treated section, both 32 diameters long. The two test sections are mounted in series, the first preceded by a 120 diameter long untreated developing section and the second preceded by a 60 diameter long SLIPS treated developing section, to ensure fully developed pipe flow. The effects of SLIPS using oils of different viscosity on the flow resistance is quantified by simultaneously measuring and comparing the pressure drop along the untreated and the SLIPS treated test sections.

[1]Supported under ONR Grants N00014-12-1-0875 and N00014-12-1-0962 (program manager Ki-Han Kim). M.L. was supported by the Lewis fund for innovation in Energy and the Environment through Princeton Andlinger Center for Energy and the Environment.

1:18PM R10.00002 Discharge Coefficients for Irregular Orifices , WADE HUEBSCH, DONALD GRAY, GREG THOMPSON, P.J. SPAUR, West Virginia University — The flow of water through an irregular orifice into an air space is of interest as a model for the flow into a submerged tunnel through a rupture of the wall. An experimental study was conducted of the flow though circular and noncircular thin plate orifices. Steady state discharge coefficients (C_d) were measured gravimetrically and photographs of the free jets were taken. Contrary to expectations, C_d increased as the shape deviated from a circle: $C_d = 0.61$ (circle), 0.625 (square), 0.654 (ellipse), and 0.665 (circular-sawtooth). Four irregular orifices had $C_d = 0.657$ (least irregular), 0.661, 0.685, and 0.704 (most irregular). The discharge coefficient for the circular orifice agreed with classical results. The jet from the square orifice showed evidence of the expected inversion from a square cross section to a cross. For the elliptical orifice, C_d was independent of the smoothness of the orifice edge. Discharge coefficients were also measured for extruded tubes of various lengths having circular and two irregular cross sections. In all cases the flow changed from a free jet to one which filled the tube at a dimensionless tube length of 2.

1:31PM R10.00003 Breaking the Symmetry with Flexible Blades: Part II , JULIA COSSE, California Institute of Technology, DAEGYOUM KIM, Brown University, LUTZ MUELLER, MORTEZA GHARIB, California Institute of Technology — Vertical axis wind turbines use various methods of asymmetry to promote rotation. Historically two main methods were used; rigid blades with complex shapes or walls blocking the wind from passing through the upwind moving half of the rotor. This project has investigated the use of flexibility as a simpler alternative with great success. A model turbine with interchangeable blades was built and tested in a wind tunnel when configured with several blades of different materials. We found that rotation occurred only when the turbine was equipped with the flexible blades. Successful wind tunnel studies motivated field-testing of the turbine. This talk addresses the recent results regarding the flexible bladed wind turbine testing in the fickle wind environment of the Caltech field laboratory for wind energy (FLOWE). This research is supported by the Gordon and Betty Moore foundation.

1:44PM R10.00004 Experimental Investigation of Scalar Patterns in a Spatially Periodic Flow Field , HADI RAJAEI, OZGE BASKAN, MICHEL SPEETJENS , HERMAN CLERCX , Eindhoven University of Technology — Spatially persisting patterns that form during the downstream evolution of passive scalars in three-dimensional (3D) spatially periodic flow fields is a fundamental aspect of industrial mixing processes that rely on the static mixing principle. Despite many numerical studies on 3D spatially periodic flow fields, a comprehensive experimental research on the periodic flow field is still scarce. This research focuses on a comparative analysis between laboratory experiments and numerical simulations on the evolution of the periodic flow field and coherent flow structures and concerns the effect of boundary (i.e. inlet) conditions on the periodicity of the flow field in a representative inline mixer consisting of spatially periodic internal elements. The experimental setup is composed of an optically accessible vertical test section with transparent internal elements which is fed by a pressure-driven pipe flow containing tracer particles. The 3D streamlines, hence, the flow field, are measured by 3D Particle-Tracking Velocimetry technique. The streamwise development of the periodicity within few internal elements regardless of the inlet conditions has been proven by preliminary studies.

1:57PM R10.00005 Leaping shampoo glides on a 500-nm-thick lubricating air layer , ERQIANG LI, King Abdullah University of Science and Technology, SANGHYUN LEE, Texas A&M University, JEREMY MARSTON, King Abdullah University of Science and Technology, ANDREA BONITO, Texas A&M University, SIGURDUR THORODDSEN, King Abdullah University of Science and Technology — When a stream of shampoo is fed onto a pool in one's hand, a jet can leap sideways or rebound from the liquid surface in an intriguing phenomenon known as the Kaye effect. Earlier studies have debated whether non-Newtonian effects are the underlying cause of this phenomenon, making the jet glide on top of a shear-thinning liquid layer, or whether an entrained air layer is responsible. Herein we show unambiguously that the jet slides on a lubricating air layer [Lee et al., Phys. Rev. E, 87, 061001 (2013)]. We identify this layer by looking through the pool liquid and observing its rupture into fine micro-bubbles. The resulting micro-bubble sizes suggest that the thickness of this air layer is around 500 nm. This thickness estimate is also supported by the tangential deceleration of the jet during the rebounding, with the shear stress within the thin air layer sufficient for the observed deceleration. Particle tracking within the jet shows uniform velocity, with no pronounced shear, which would be required for shear-thinning effects. The role of the surfactant may primarily be to stabilize the air film.

2:10PM R10.00006 A new effective macroscopic Stokes-Cahn-Hilliard formulation for immiscible fluids in porous structures , MARKUS SCHMUCK, MARC PRADAS, Department of Chemical Engineering, Imperial College London, UK, GRIGORIOS PAVLIOTIS, Department of Mathematics, Imperial College London, UK, SERAFIM KALLIADASIS, Department of Chemical Engineering, Imperial College London, UK — Guided by thermodynamic and variational principles we describe mixtures of incompressible fluids in strongly perforated domains with a general class of phase field equations coupled to the Stokes equations. Important applications include subsurface flows, fuel cells, and microfluidics. Starting with a microscopic formulation for heterogenous domains (e.g. a porous medium), represented as the periodic covering of a single reference cell, we rigorously derive effective macroscopic phase field equations under the assumption of periodic flow for large Péclet numbers by the multiple scale method with drift and our recently introduced splitting strategy for Ginzburg-Landau/Cahn-Hilliard-type equations [1]. We recover systematically diffusion-dispersion relations (including Taylor-Aris-dispersion) as for classical convection-diffusion problems. Our results provide a convenient computational framework to macroscopically track interfaces in porous media. In view of the well-known versatility of phase field models, our study proposes a promising formulation for many engineering and scientific applications such as multiphase flows in porous media and oil recovery, for instance.

[1] Schmuck, Pradas, Pavliotis, Kalliadasis, 2012. Proc. R. Soc. A 468, 3705 (2012).

2:23PM R10.00007 Spin-down of a rotating air hockey disk , PATRICK WEIDMAN, KEITH JULIEN, University of Colorado — We extend the work of Weidman (APS, DFD 2008) on the steady float height of a rotating disk to formulate and solve for the unsteady behavior of spin-down to rest. A similarity reduction of the Navier-Stokes equations reduces the problem to a coupled pair of partial differential equations in space and time. For a disk of fixed radius and density, the PDE's must be solved taking into account constraints imposed by the aerodynamic torque and aerodynamic lift. Thus the full solution for the unsteady azimuthal and axial dynamics of the disk can be obtained for given initial values of disk Reynolds number $R = W\,h/\nu$ and dimensionless disk rotation speed $S = \sqrt{2}\Omega h/W$, where h is the float height, W is the fluid levitation velocity, Ω is the disk rotation rate, and ν is the kinematic viscosity of the fluid. Integrations reveal interesting families of solutions when plotted over steady solution curves in R-S parameter space and vindicate the quasi-steady spin-down theory reported in earlier work, valid only in a restricted region of parameter space.

2:36PM R10.00008 Experimental Characterization of Inter-channel Mixing Through a Narrow Gap , SIMO A. MAKIHARJU, Department of Mechanical Engineering, University of Michigan, Ann Arbor, MI 48109, ALEXANDER G. MYCHKOVSKY, JOHN R. BUCHANAN JR., KEVIN J. HOGAN, KIRK T. LOWE, Bechtel Marine Propulsion Corporation, Bettis Atomic Power Laboratory, West Mifflin, PA 15122, STEVEN L. CECCIO, Department of Naval Architecture and Marine Engineering, University of Michigan, Ann Arbor, MI 48109 — Mixing trough narrow gaps that connect primary flow paths is an important flow process for many thermal-hydraulic applications, such as flow through nuclear reactor rod bundles or heat exchangers. The flow in a narrow gap can exhibit periodic flow structures due to travelling vortices. These flow structures in the gap, as well as any pressure gradient across the gap, have a significant effect on the rate of mixing between the primary flow paths. To investigate such flows in detail, and to develop validation quality data sets for comparison with CFD, we have conducted a canonical inter-channel mixing experiment between two channels, with a (127 mm)^2 cross-section. The channels were connected by a gap 914.4 mm long in the stream-wise direction and 50.8mm wide in the cross-stream direction. The gap height could be varied from 0 to 50.8 mm. The flow speed in both channels could be independently varied to have Re = (40 to 100) x 10^3. The integral mixing rates were determined by injecting fluorescent dye into one of the channels well upstream of the test section and by measuring the dye concentration at the channel inlets and outlets. Additionally, the flow fields in the gap and channels were measured with LDV and PIV.

2:49PM R10.00009 Investigations of Flow past Spinning Cylinders[1] , IGBAL MEHMEDAGIC, PASQUALE CARLUCCI, LIAM BUCKLEY, DONALD CARLUCCI, U. S. Army ARDEC, Picatinny Arsenal, NJ, ELIAS ALJALLIS, SIVA THANGAM, Stevens Institute of Technology, Hoboken, NJ — A subsonic wind tunnel is used to perform experiments on flow past spinning cylinders. The blunt cylinders are sting-mounted and oriented such that their axis of rotation is aligned with the mean flow. The experiments cover a Reynolds number range of up to 300000 and rotation numbers of up to 1.2 (based on cylinder diameter). The results for spinning cylinders with both rear-mounted and fore-mounted stings are presented. Computations are performed using a two-equation anisotropic turbulence model that is based on proper representation of the energy spectrum to capture rotation and curvature. The model performance is validated with benchmark experimental flows and implemented for analyzing the flow configuration used in the experimental study.

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

3:02PM R10.00010 The High Reynolds Number Limit of Nonlinear Equilibrium States in Couette Flow , KENGO DEGUCHI, PHILIP HALL, Imperial College London — In recent years there has been much interest in the nonlinear equilibrium states, which Couette flow can support. Computational interest in this problem began with Nagata in the early 1990's and since then there have been numerous investigations of shear flows. Beginning in the late 1980's Hall and Smith laid down an asymptotic framework, referred to as vortex-wave interaction theory, for nonlinear equilibrium states generated by the interaction of TS or inviscid wave systems in a variety of shear flows. In 2010 Hall and Sherwin showed conclusively that the asymptotic theory described what were referred to in the CFD community as self sustained processes. Subsequently Deguchi, Hall and Walton (2013) showed that the asymptotic states at small wavenumbers took on a new structure capable of describing spot formation. The asymptotic theory was however unable to predict the so-called upper branch self-sustained process found numerically and the theory predicted a critical wavenumber beyond which asymptotic states could not be found. Here the upper branch structure is given in the high Reynolds number limit and the development of asymptotic modes beyond the critical wavenumber is given thereby completing the description of equilibrium states at high Reynolds numbers.

3:15PM R10.00011 Nonlinear Equilibrium States in Growing Boundary Layers , PHILIP HALL, KENGO DEGUCHI, Imperial College — Recently there has been much interest in vortex wave interaction/self sustained process/coherent structures in fully developed flows. In growing boundary layers the local Reynolds number varies in the flow direction and so the relevance of equilibrium states calculated for developed flows is in doubt. Here results are presented for nonlinear states in quite general boundary layers. Some of the structures we find using asymptotic and numerical methods are "distant cousins" of structures found in for example Couette flow whilst others are apparently unrelated. The new states are completely dependent on the background state being a boundary layer and are found have an elegant canonical asymptotic form. The new states are shown to be valid for quite general boundary layers; in addition they are related to experimental observations.

Tuesday, November 26, 2013 1:05PM - 2:49PM –

Session R11 Non-Newtonian Flows III 335 - Eric Johnsen, University of Michigan

1:05PM R11.00001 Numerical simulation of particle migration in rotating eccentric cylinders , BRIAN PALMER, DIWEN MENG, PARISA MIRBOD, Clarkson University — In this study we numerically investigate the particle migration in a concentrated suspension undergoing flow between rotating eccentric cylinders observed in an experiment by Subia et al. (1998) J. Fluid Mech., 373. There are two mathematical models developed to explain particle migration phenomenon, namely, suspension balance model and diffusive flux model. These models have been successfully applied to explain migration behavior in several two-dimensional flows. However, compared with two-dimensional simulation, three-dimensional simulation is able to produce relatively realistic results. In this work, we have carried out numerical simulation of concentrated suspension flow in 3D eccentric cylinder geometry using suspension balance model and finite element methods. The simulation method was validated with available analytical solutions for circular Couette flow. Therefore, the simulation technique was applied to study the flow of concentrated suspensions through rotating eccentric cylinders. The detailed comparison of numerical simulation results is made with the experimental data.

1:18PM R11.00002 Coarse-grained simulations of flow-induced morphology dynamics in dispersed graphene , YUEYI XU, MICAH GREEN, Department of Chemical Engineering, Texas Tech University — We investigated how flow fields affect graphene morphology dynamics in liquid phase using a coarse-grained model. Past simulations of the dynamics of dispersed graphene sheets are limited to static fluids on small timescales, with little attention devoted to flow dynamics, which is critical given the importance of graphene solution-processing of multifunctional devices and materials. We developed a Brownian Dynamics (BD) algorithm to study the morphology of sheetlike macromolecules in dilute solutions with an applied external flow field. We used a bead-rod lattice to represent the mesoscopic conformation of individual two dimensional sheets. We then analyzed the morphology dynamic modes (stretching, tumbling, crumpling) of these molecules as a function of sheet size, Weissenberg number, and bending stiffness. The physical properties (e. g. viscosity) affected by the morphology are also studied. Our results demonstrate how bending stiffness relates to relaxation modes during startup of shear.

1:31PM R11.00003 Stretch and relax: a viscoelastic filament that displays thixotropic yield stress behavior[1] , YURIKO RENARDY, HOLLY GRANT, Virginia Tech — A filament with circular cross-sectional area is stretched by controlling the distance between the ends, and then stopped. The evolution of the filament radius in the presence of gravity and surface tension is studied. The constitutive model is a combination of a Newtonian solvent and the viscoelastic partially extending strand convection model (Larson 1986) with a large relaxation time. Time-dependent solutions show phenomena with thixotropy and yield stress.

[1]This research is supported by NSF-DMS.

1:44PM R11.00004 Ants cushion applied stress by active rearrangements , ZHONGYANG LIU, JOHN HYATT, NATHAN MLOT, MICHAEL GEROV, ALBERTO FERNANDEZ-NIEVES, DAVID HU, Georgia Institute of Technology — Fire ants, Solenopsis invicta, link their bodies together to form waterproof rafts, which in turn drip, spread, and coagulate, demonstrating properties of an active material that can change state from a liquid to a solid. This soft-matter phase transition is important when the raft interacts with environmental forces such as raindrops and crashing waves. We study this active behavior through plate-on-plate rheology on the ants, extracting the active components by comparison with the rheological behavior of a collection of dead ants. In controlled shear tests, both and live and dead ants show properties of a non-Newtonian fluid, specifically, shear-thinning behavior. In oscillatory tests, live ants exhibit a rare behavior in which their storage modulus (G') and loss modulus (G") have approximately the same value over three orders magnitudes of frequency and two orders of magnitude of strain, indicating the ants are neither fluid nor solid. In comparison, dead ants are more solid-like, with a storage modulus twice as large as their loss modulus. This striking active behavior arises from rearrangement of their bodies and storage and dissipation of energy with the ants' muscles.

1:57PM R11.00005 Simulations of Shock Propagation in Viscoelastic Media , MAURO RODRIGUEZ, ERIC JOHNSEN, University of Michigan, Ann Arbor — Understanding the mechanics of shock waves emitted by cavitation bubbles and propagating through viscoelastic media is important to various naval and medical applications, particularly in the context of cavitation damage. In such problems, the constitutive models describing the material are non-trivial, and include effects such as nonlinear elasticity, history and viscosity. Thus, the influence of the shock on the material and the response of the material to the shock are generally unknown. A novel numerical approach is proposed for simulating shock and acoustic-wave propagation in a Zener-like viscoelastic medium. The method is based on a high-order accurate weighted essentially non-oscillatory (WENO) scheme for shock capturing and introduces evolution equations for the stresses. The HLLC Riemann solver is used for upwinding, with a reconstruction of the primitive variables. The performance and accuracy of the numerical approach is presented for several one- and two-dimensional problems, including acoustic wave propagation and the Sod shock tube problem for various combinations of elasticities, viscosities and relaxation times. This work is supported by ONR grant N00014-12-1-0751.

2:10PM R11.00006 Viscoelastic Effects on Spraying and Fragmentation of Polymeric Solutions , BAVAND KESHAVARZ, GARETH MCKINLEY, MIT, Mech. Eng. Dept., ERIC HOUZE, JOHN MOORE, MICHAEL KOERNER, Axalta, MIT, MECH.,ENG.,DEPT. TEAM, AXALTA COATING SYSTEMS TEAM — The addition of small amounts of polymer to Newtonian fluids can inhibit the spray process, but the physical reasons behind these effects are still unclear. To explore this phenomenon, model viscoelastic fluids composed of very dilute solutions of polyethylene oxide are tested in a variety of fragmentation processes including air-assisted atomization, jet impact fragmentation, drop impact, and rotary atomization. Spray image analysis shows that when the fluid viscoelasticity is increased the average particle diameter and Sauter Mean Diameter both show a systematic increase before reaching an asymptotic plateau value. As observed for Newtonian fluids, the droplet size distributions are still well described by a Gamma distribution but the addition of viscoelasticity shifts the distribution to smaller values of n, corresponding to a broader size distribution. A linear stability analysis indicates that the effects of fluid viscoelasticity are more pronounced in the final stage of ligament formation than in the initial stages of atomization. The linear analysis can predict the observed trends in the mean droplet sizes; however, the shift in the size distributions seems to rise from the nonlinear dynamics of the stretched viscoelastic ligaments close to break up.

2:23PM R11.00007 Elasto-Inertial Turbulence in polymeric flows , VINCENT TERRAPON, University of Liege, YVES DUBIEF, University of Vermont, JULIO SORIA[1], Monash University — The dynamics of elasto-inertial turbulence (EIT) is investigated numerically from the perspective of the coupling between polymer dynamics and flow structures. In particular, direct numerical simulations of channel flow with Reynolds numbers ranging from 1000 to 6000 are used to study the formation and dynamics of elastic instabilities and their effects on the flow. Based on the splitting of the pressure into inertial and polymeric contributions, it is shown that the trains of cylindrical structures around thin sheets of high polymer extension that are characteristics to elasto-inertial turbulence are mostly driven by polymeric contributions.

[1]Also King Abdulaziz University, Jeddah

2:36PM R11.00008 A numerical rheometer, or the time-dependent flow of dense suspensions[1] , GUSTAF MÅRTENSSON, Chalmers University of Technology, Sweden, Micronic Mydata AB, Sweden, ANDREAS MARK, FREDRIK EDELVIK, Fraunhofer-Chalmers Research Centre, Sweden, LARS ESSÉN, Micronic Mydata AB, Sweden — The ability to perform numerical simulations of realistic flow situations is an ongoing area of research of importance in the realm of academia, as well as industry. The goal of this project is to perform a proof-of-concept calculation on the measurement of the rheological properties for a dense suspension. The following measurement methods were chosen for evaluation: a) Couette rheometry, b) parallel plate rheometry and c) axial piezo-rheometry. In the case of Couette and parallel plate rheometry, the rotational motion of the cylinder or plate is controlled with respect to the rotational rate or induced strain, whereafter a torsional moment is measured. For the piezo-rheometry, the stress mode is in the normal direction with respect to the plate and a stress response is measured. Experimental data for a number of model fluids is used for validation The simulations are performed with IBOFlow, the multi phase flow solver developed at Fraunhofer-Chalmers Centre. The granular-suspension is modelled by a two-fluid model discretized in an Euler-Euler framework. From the simulations it is concluded that the proposed granular model accurately models the rheology of the general flux and that the relaxation time may vary from case to case.

[1]Supported by the Swedish Research Council

Tuesday, November 26, 2013 1:05PM - 3:02PM –

Session R12 Vortex Dynamics and Vortex Flows X 336 - Megan Leftwich, George Washington University

1:05PM R12.00001 Volumetric Velocity Fields Downstream of a 2-Bladed Turbine , DANIEL TROOLIN, TSI Inc. Fluid Mechanics Research Instruments — Tip vortices of axial-flow turbines are important in understanding the mean and turbulent characteristics of the wake. Volumetric 3-component velocimetry (V3V) was used to examine the flow downstream of a model two-bladed turbine in air. The turbine had a diameter of 177.8 mm and was powered by a motor operating at approximately 150 rpm. The measurement volume (50 x 50 x 20 mm) was positioned approximately 5 mm downstream of the blade tip, in order to examine the tip vortex structure. The V3V system utilized three 4MP cameras with 85mm lenses positioned in a fixed triangular frame located at a distance of 450 mm from the back of the measurement volume. The illumination source was a 200 mJ dual-head pulsed Nd:YAG laser operating at 7.25 Hz and illuminating 1 micron olive oil droplets as tracer particles. The particle images were then analyzed to produce volumetric vector fields. The focus was placed on visualizing the complex interaction between the turbine tip vortices. Insights on the tip vortex dynamics and three dimensional characteristics of the wake flow will be discussed.

1:18PM R12.00002 Large-eddy simulations of a single vertical axis wind turbine , MAHSA RAHRO-MOSTAQIM, ANTONIO POSA, ELIAS BALARAS, MEGAN LEFTWICH, The George Washington University — Recently vertical axis wind turbines (VAWTs) have been receiving increased attention due to various potential advantages over the more common horizontal axis wind turbines. They can be placed for example in urban areas where space is limited, since they are moderately sized and virtually silent. In this study we will report large-eddy simulations (LES) of a Windspire VAWT. Computations will be conducted using an immersed boundary formulation, where the equations of motion are solved on a fixed Cartesian grid and the turbine blades rotate with a fixed tip speed ratio. The primary objective of this first series of LES is to understand the interaction between the wakes generated by the individual airfoils. To keep the computational cost low and increase the parametric regime we can examine, we will consider only part of the turbine hight and utilize periodic boundary conditions along the turbine axis. The computations will exactly mimic the conditions of closely coordinated experiments of a scaled down VAWT, which will enable us to access the impact of features that will not be captured, such as the tip vortices for example, on the results. Preliminary results reveal a complex interaction of the wakes created by the rotating airfoils and the boundary layer on the airfoils.

1:31PM R12.00003 An Actuator Curve Embedding Method to Model Wind Turbine Wakes , PANKAJ JHA[1], SVEN SCHMITZ[2], The Pennsylvania State University — The Actuator Line Method (ALM) is widely used in the wind energy community to model the complex interactions within large wind farms in large-eddy simulation (LES) of the atmospheric boundary layer (ABL) at various stability states. The state-of-the-art in ALM modeling is rooted in the work of Sorensen and Shen (2002). The major weakness of the ALM still remains in having the actuator line discretized as a superposition of individual spherically-spread body forces. The associated overlap of adjacent spherical force fields leads to a large sensitivity of computed blade loads to the way in which the spherical spreading radius is altered along the actuator line (Jha et al. 2013). An Actuator Curve Embedding (ACE) method is developed that considers a general actuator line in 3-D space where the force distribution along the actuator curve is embedded continuously onto the background mesh and without overlap. The ACE method thus is expected to show improved body-force discretization for wind turbine blades, in particular those subject to aeroelastic deformations. Some preliminary results contrasting the ALM and ACE methods are discussed. *Support: DOE.*

[1]Graduate Research Assistant, Aerospace Engineering
[2]Assistant Professor, Aerospace Engineering

1:44PM R12.00004 Three-dimensional velocity measurements around a rotating vertical axis wind turbine , FILIPPO COLETTI, KEVIN RYAN, Stanford University, JOHN DABIRI, California Institute of Technology, JOHN EATON, Stanford University — Vertical axis wind turbines (VAWT) can be more closely spaced than conventional horizontal axis wind turbines (HAWT), which points to a potentially greater power that can be extracted from a given wind farm footprint. In order to optimize the inter-turbine spacing and to investigate the potential for constructive aerodynamic interactions, the complex dynamics of VAWT wakes need to be analyzed. To date, only single-point or at best two-dimensional measurements of such wakes have been documented. We have measured the full three-component mean velocity field around and downstream the scaled-down model of a rotating VAWT by Magnetic Resonance Velocimetry (MRV). The high spatial resolution allows to quantitatively explore the structure of the wake, its interaction with the floor, and its development. The flow is shown to be highly three-dimensional and asymmetric for the whole investigated region (up to 7 diameters downstream of the turbine). These results can inform low-order models to predict the performance of turbine arrays.

1:57PM R12.00005 The wake of a single vertical axis wind turbine , DANIELLE A. BARSKY, MEGAN C. LEFTWICH, George Washington University — The purpose of this study is to measure the wake of a Windspire vertical axis wind turbine (VAWT). In recent years, research on VAWTs has increased due to various potential advantages over the more common horizontal axis wind turbines (HAWTs). Unlike very large HAWTs, moderately sized–and virtually silent–VAWTs can be placed in urban and suburban regions where land space is limited. To date, many VAWT studies have assumed that the turbine has the same aerodynamic structure as a spinning cylinder despite a significant increase in geometric complexity. This experiment attempts to understand the fundamental wake structure of a single VAWT (and compare it to the wake structure of a spinning cylinder). In this experiment, a scaled-down VAWT is placed inside a wind tunnel under a controlled laboratory setting. A motor rotates the scale model at a constant angular speed. Stereo particle image velocimetry (PIV) is used to visualize the wake of the turbine and image processing techniques are used to quantify the velocity and vorticity of the wake.

2:10PM R12.00006 Experimental investigation of the wake characteristics of flow-powered and motorized laboratory-scale wind turbines[1] , DANIEL ARAYA, JOHN DABIRI, California Institute of Technology — We present experimental data that compares the wake characteristics of a laboratory-scale vertical-axis turbine while it is either powered by the flow or by a DC motor. This distinction is relevant for laboratory experiments in which scale turbine models are used that require the use of a motor to spin the turbine blades. Particle image velocimetry is used to measure the velocity field in a two-dimensional plane normal to the axis of rotation. This velocity field is then used to compare time-averaged streamwise velocity, turbulence kinetic energy, and power of the two configurations. The results give insight into the kinematic effect of adding energy to the flow by way of the motor, and they suggest limits on the extrapolation of laboratory results to full-scale performance.

[1]This work was supported by an NSF Graduate Research Fellowship to D.B.A and funding to J.O.D. from ONR N000141211047.

2:23PM R12.00007 Experimental Study of Vortex Dynamics during Blade-Vortex Interactions , DI PENG, JAMES GREGORY, The Ohio State University — Vortices incident upon bodies, such as cylinders, airfoils, and rotor blades, can give rise to substantial unsteady loading, sound generation, and vibration in a variety of engineering applications. A comprehensive study on vortex dynamics during blade-vortex interaction (BVI) is performed in this work. Evidence has been found in previous studies that the vortex behavior during BVI varies with Reynolds number, but the effects are not clear. In the current study, the experiments are performed in a 3' $\times$ 5' low speed wind tunnel where the Reynolds number can be varied from 6×10^4 to 8×10^5 by adjusting freestream speed and airfoil size. The vortex is generated by the pitching motion of a wing, which is driven by an air cylinder. Another wing is placed downstream to initiate parallel interactions with the generated vortices. Smoke visualization is used originally to characterize the vortex. Then the BVI problem is studied in detail using time-resolved PIV and unsteady pressure measurements on the downstream target airfoil. The vortex behaviors at selected Reynolds numbers are investigated. The influence of other factors on vortex behavior, such as vortex strength and core size, is also discussed.

2:36PM R12.00008 Combined Vorticity Confinement and Total Variation Diminishing Technique for Modeling of Blade Tip Vortex[1] , ALEX POVITSKY, KRISTOPHER PIERSON, University of Akron — The Vorticity Confinement (VC) approach is combined with Total Variation Diminishing (TVD) technique to avoid over-confinement and divergence of upwind second-order of approximation schemes. The TVD schemes were combined with the first (constant confinement parameter ε) and second (constant unit-less confinement parameter c) VC formulations and with adoptive VC formulation by Hahn and Iaccarino. Combined VC/TVD techniques were first applied to convected Taylor vortex, which represent a model of wing tip vortex. For the former two VC methods combination of the second-order upwind discretization scheme with VC shows significant over-confinement of vortex whereas the first-order discretization scheme leads to strong dissipation of vortex. While the latter VC technique shows acceptable results for first-order upwind scheme, it either diverges or strongly over-confines when the second-order upwind discretization scheme is used. The VC/TVD techniques were tested with non-differentiable minmod and Van Leer flux limiters and with differentiable Van Albada limiter. The combination of VC and TVD with differential limiter computes most accurate vortex. The proposed technique is applied to tip vortex generated by rotating blade. Implementation of combined VC with TVD equipped with differential flux limiter to CFD code FLUENT shows much more close comparison to experimental results in terms of vortex velocity profile and size of vortex core compared to the same CFD code without VC approach.

[1]Grant support from ARO, AFRL, DAGSI

2:49PM R12.00009 ABSTRACT WITHDRAWN –

Tuesday, November 26, 2013 1:05PM - 3:02PM –

Session R13 Granular Flows V: Fluctuations and Instabilities 301 - Bob Behringer, Duke University

1:05PM R13.00001 Granular Impact: Predicting Dynamics with a Collisional Model[1] , ABE CLARK, ALEC PETERSEN, ROBERT BEHRINGER, Duke University — Granular impact, where a high-speed intruder strikes a granular material, occurs frequently in industrial and natural processes. Due in part to difficulty in obtaining sufficiently fast data at the grain scale, a complete description, which connects grain-scale processes to macroscopic dynamics of the intruder, is still lacking. Using photoelastic particles and a high speed camera, we perform experiments which allow us to capture the intruder dynamics and local granular force response at very fast time scales. This allows us to approach the problem both macroscopically, fitting the intruder dynamics to a fluid-like force law which is dominated by a velocity-squared drag force, and microscopically, where we observe large force fluctuations at small space and time scales. Thus, the intruder deceleration is not smooth and steady, but dominated by intermittent collisions with clusters of grains. Based on this, we present a simple collisional model which yields a shape-dependent velocity-squared drag force. Using experimental data, we show that this model captures the shape-dependence of the intruder deceleration and o-axis rotation well. This confirms the microscopic assumptions of this model, and may provide insight into other dense, driven granular flows.

[1]Supported by DTRA, grant HDTRA1- 10-0021

1:18PM R13.00002 Instability in shocked granular gases , NICK SIRMAS, MATEI RADULESCU, University of Ottawa — Shocks in granular media, such as vertically oscillated beds, have been shown to develop instabilities. Similar jet formation has been observed in explosively dispersed granular media. In the current study, we investigate the origin of this instability. Our previous work addresses this instability by performing discrete-particle simulations of inelastic media undergoing shock compression. By allowing finite dissipation within the shock wave, instability manifests itself as distinctive high density non-uniformities and convective rolls within the shock structure. By analyzing the time evolution of the material undergoing the shock wave compression and further relaxation, we found that the clustering instability is the dominant mechanism controlling this instability. In the present study we extend this work to investigate the instability at the continuum level. We model the Euler equations for granular gases with a modified cooling rate to include an impact velocity threshold necessary for inelastic collisions. Our results demonstrate a fair agreement between the continuum and discrete-particle models. Slight discrepancies, such as higher frequency non-uniformities in our continuum results may be attributed to the absence of viscous effects.

1:31PM R13.00003 Linear and nonlinear response in sheared soft spheres , BRIAN TIGHE, Delft University of Technology — Packings of soft spheres provide an idealized model of foams, emulsions, and grains, while also serving as the canonical example of a system undergoing a jamming transition. Packings' mechanical response has now been studied exhaustively in the context of "strict linear response," i.e. by linearizing about a stable static packing and solving the resulting equations of motion. Both because the system is close to a critical point and because the soft sphere pair potential is non-analytic at the point of contact, it is reasonable to ask under what circumstances strict linear response provides a good approximation to the actual response. We simulate sheared soft sphere packings close to jamming and identify two distinct strain scales: (i) the scale on which strict linear response fails, coinciding with a topological change in the packing's contact network; and (ii) the scale on which linear superposition of the averaged stress-strain curve breaks down. This latter scale provides a "weak linear response" criterion and is likely to be more experimentally relevant.

1:44PM R13.00004 Capillary-like Fluctuations of a Solid-Liquid Interface in a Non-Cohesive Granular System[1] , NICOLAS MUJICA, LI-HUA LUU, GUSTAVO CASTILLO, RODRIGO SOTO, Departamento de Fisica, FCFM, Universidad de Chile — One of the most noticeable collective motion of non-cohesive granular matter is clustering under certain conditions. In particular, when a quasi-two-dimensional monolayer of mono-disperse non-cohesive particles is vertically vibrated, a solid-liquid-like transition occurs when the driving amplitude exceeds a critical value. Here, the physical mechanism underlying particle clustering relies on the strong interactions mediated by grain collisions, rather than on grain-grain cohesive forces. In average, the solid cluster resembles a drop, with a striking circular shape. We experimentally investigate the coarse-grained solid-liquid interface fluctuations, which are characterized through the static and dynamic correlation functions in the Fourier space. These fluctuations turn out to be well described by the capillary wave theory, which allows us to measure the solid-liquid interface surface tension and mobility once the granular thermal kinetic energy is determined. Despite the system is strongly out of equilibrium and that the granular temperature is not uniform, there is energy equipartition at the solid-liquid interface, for a relatively large range of angular wave-numbers. Furthermore, both surface tension and mobility are consistent with a simple order of magnitude estimation.

[1]Fondecyt Grants No. 3120172 (L.-H.L), No. 1120211 (G.C. & N.M.) and No. 1100100 (R.S.), and Anillo ACT 127 and AIC 43

1:57PM R13.00005 Nonlinear bounded convection and a phase separation instability in a dilute granular gas , PRIYANKA SHUKLA, Indian Institute for Science Education and Research, Kolkata, India, MEHEBOOB ALAM, Jawaharlal Nehru Centre for Advanced Scientific Research, Bangalore 560064, India — A weakly nonlinear analysis using Stuart-Landau equation has been carried out to understand the onset of buoyancy-induced convection and a phase-separation instability of a dilute granular gas in a bounded domain. Previous linear stability analysis of the same base state showed that the conduction state is unstable for a range of Froude number and heat-loss parameter. A new instability mode has been found at very small values of the heat loss parameter, the origin of which is shown to be connected to the classical Rayleigh-Benard instability. The bifucation diagrams and the supercritical and subcritical nonlinear patterns with respect to inelasticity and Froude number will be discussed.

2:10PM R13.00006 Numerical simulation of unsteady chute flows of two-phase granular mixtures , CHRISTOS VARSAKELIS, MILTIADIS V. PAPALEXANDRIS, Université catholique de Louvain — The unsteady gravity-driven flow of a fluid-saturated granular material on an inclined plane is investigated numerically. Our studies are based on a continuum two-phase flow model for the mixtures of interest. The governing equations are integrated via a predictor-corrector algorithm that employs a generalized projection method for the computation of the phasial pressures. Further, it incorporates an interface detection and capturing method to account for the steep gradients of particle concentration across material interfaces. In our numerical setup, a dense granular layer of constant thickness is placed on the surface of an inclined plane, whereas the rest of the domain is filled with an interstitial fluid. Initially the mixture is assumed to be at rest and is accelerated by gravity. A representative sample of these simulations is presented and discussed. Since the flows of interest are susceptible to Kapitza instability, emphasis is placed on the spatio-temporal evolution of the granular layer's free surface and the interplay between inertia and gravity. Also, we discuss the flow characteristics inside the granular layer and we compare the predicted profiles for the phasial variables with those obtained from previous studies.

2:23PM R13.00007 Spatially modulated kinks in shallow granular layers[1] , CLAUDIO FALCÓN, JUAN MACÍAS, Departamento de Física, Facultad de Ciencias Físicas y Matemáticas, Universidad de Chile, Casilla 487-3, Santiago, Chile — We report on the experimental observation of spatially modulated kinks in a shallow one-dimensional fluidized granular layer subjected to a periodic air flow. We show the appearance of these solutions as the layer undergoes a parametric instability. Due to the inherent fluctuations of the granular layer, the kink profile exhibits an effective wavelength, *a precursor*, which modulates spatially the homogeneous states and drastically modifies the kink dynamics. We characterize the average and fluctuating properties of this solution. Finally, we show that the temporal evolution of these kinks is dominated by a hoping dynamics, related directly to the underlying spatial structure.

[1]FONDECYT No. 1130354

2:36PM R13.00008 ABSTRACT WITHDRAWN –

2:49PM R13.00009 Dynamic X-ray study of the effective temperature in a three-dimensional granular gas , YUJIE WANG, BINGQUAN KOU, HAOHUA SUN, YIXIN CAO, CHENGJIE XIA, XIAODAN ZHANG, Department of Physics, Shanghai Jiao Tong University, 800 Dong Chuan Road, Shanghai 200240, China, XIANGHUI XIAO, KAMEL FEZZAA, X-ray Science Division, Argonne National Laboratory, 9700 South Cass Avenue, IL, USA — We carried out a high-speed x-ray imaging study of the effective temperature of a highly agitated three-dimensional (3D) granular gas in the tracer limit using tracing particles with various densities, restitution coefficients, and sizes. Both the tracing and background particles satisfy non-Gaussian velocity distributions, in addition to an absence of energy equipartition between translational and rotational degrees of freedom.

Tuesday, November 26, 2013 1:05PM - 2:49PM –

Session R14 Rotating Flows II 302 - Stephan Weiss, University of Michigan

1:05PM R14.00001 Boundary layers and global stability of laboratory quasi-Keplerian flow , E.M. EDLUND, H. JI, Princeton Plasma Physics Laboratory — Studies in the HTX device at PPPL, a modified Taylor-Couette experiment, have demonstrated a robust stability of astrophysically relevant, quasi-Keplerian flows. Independent rings on the axial boundary can be used to fine tune the rotation profile, allowing ideal Couette rotation to be achieved over nearly the entire radial gap. Fluctuation levels in these flows are observed to be at nearly the noise floor of the laser Doppler velocimetry (LDV) diagnostic, in agreement with prior studies under similar conditions. Deviations from optimal operating parameters illustrate the importance of centrifugally unstable boundary layers in Taylor-Couette devices of the classical configuration where the axial boundaries rotate with the outer cylinder. The global stability of nearly ideal-Couette flows, with implications for astrophysical systems, will be discussed in light of the global stability of these flows with respect to externally applied perturbations of large magnitude.

1:18PM R14.00002 Hydrodynamic turbulence in quasi-Keplerian rotating flows?[1] , LIANG SHI, Max Planck Institute for Dynamics and Self-Organization, MARC AVILA, Friedrich-Alexander-Universität ErlangenNrnberg, BJOERN HOF, The Institute of Science and Technology Austria, LIANG SHI TEAM, MARC AVILA TEAM, BJOERN HOF TEAM — The origin of turbulence in astrophysical accretion discs has been under scrutiny for decades and remains still unclear. The velocity profiles of discs (Keplerien profiles) are centrifugally stable and therefore a different instability mechanism is required for turbulence to arise. While in hot discs turbulence can be triggered through magnetorotational instability, cooler discs lack sufficient ionization and it is unclear how turbulence sets in. In analogy to other linearly stable flows like pipe and Couette flow, subcritical transition to turbulence may be the mechanism. Recently, experimental studies of Taylor-Couette flow in quasi-Keplerian regime have given conflicting results and numerical simulations of above experimental flows showed that the top and bottom end-wall leads to strong deviations from the Keplerian velocity profile and drives turbulence. In order to clarify this, we perform direct numerical simulations of incompressible Taylor-Couette flow without end walls in the quasi Keplerian regime for Re up to 200000. Strong transient growth is observed and gives rise to strongly disorted motion, suggesting that for large enough Re this mechanism may lead to turbulence even for Keplerian flows.

[1]This work is supported by Deutsche Forschungsgemeinschaft (DFG) under project SFB 963 and Max Planck Society.

1:31PM R14.00003 Geometrical statistics of the vorticity vector in rotating turbulence , HERMAN J.H. CLERCX, Phys. Department, Eindhoven University of Technology, Eindhoven, The Netherlands, LORENZO DEL CASTELLO, CNR, UOS Sapienza, Roma, Italy — We report results on the geometrical statistics of the vorticity vector obtained from particle tracking experiments in electromagnetically forced rotating turbulence. A range of rotation rates is considered, from non-rotating to rapidly rotating turbulence. Based on the full set of velocity derivatives, measured in a Lagrangian way by 3D Particle Tracking Velocimetry, we have been able to quantify statistically the effect of system rotation on several flow properties. We have studied the orientation of the vorticity vector with respect to the three eigenvectors of the local strain rate tensor and with respect to the vortex stretching vector. Additionally, we have quantified the role of system rotation on the self-amplification terms of the enstrophy and strain rate equations and the direct contribution of the background rotation on these evolution equations. The main effect of background rotation is the strong reduction of extreme events and related (strong) reduction of the skewness of PDFs of several quantities such as, for example, the intermediate eigenvalue of the strain rate tensor and the enstrophy self-amplification term. These results reflect the two-dimensionalisation of the flow at the highest rotation rates.

1:44PM R14.00004 Torque scaling and number of states in turbulent Taylor-Couette flow[1] , JORGE PEIXINHO, BORJA MARTINEZ-ARIAS, INNOCENT MUTABAZI, Le Havre University and CNRS, LABORATOIRE ONDES ET MILIEUX COMPLEXES TEAM — Torque measurements in a Taylor-Couette flow over a range of velocities up to a Reynolds number of 16 000 are presented. Here only the inner cylinder is rotating, the radius ratio is 0.9 and the aspect ratio is 30. Simultaneously to the torque, the evolution of the flow pattern is observed. Different states are observed depending on the range of Reynolds. The relationship between the states, the speed and the torque is studied in the form of scaling laws. The effect of the number of vortices and the meaning of the exponents will be discussed. In addition to Newtonian fluids, polymer solutions are also used. Specifically, the properties of low concentrations of high molecular weight poly-ethylene-oxide in water will be reported. The effect of the additives to the flow patterns and the torque scaling will be discussed.

[1]The authors acknowledge the financial support of the french Agence Nationale de la Recherche (ANR), through the program Investissements d'Avenir (ANR-10-LABX-09-01), LabEx EMC3.

1:57PM R14.00005 Turbulence decay towards the linearly-stable regime of Taylor Couette , RODOLFO OSTILLA MONICO, Physics of Fluids, Mesa+ Institute, University of Twente, SIEGFRIED GROSSMANN, Department of Physics, University of Marbuurg, ROBERTO VERZICCO, Dipartamento de Ingegneria Meccanica, University of Rome "Tor Vergata", DETLEF LOHSE, Physics of Fluids, Mesa+ Institute, University of Twente — DNS of turbulence decay in Taylor-Couette flow in the linearly stable regime is presented. A fixed radius ratio $\eta = 0.714$ is used, and initial Reynolds numbers of up to $Re \sim 10^5$ are reached. Simulations are run in an axially-periodic domain, and thus completely lack the end-plates effects which are present in experiments and cause Ekman effects leading to a supercritical transition to turbulence. Here, we start with a fully turbulent state in the unstable regime and enter the linearly stable regime by suddenly starting a (stabilizing) outer cylinder rotation. This stabilization causes the system to behave as a damped oscillator and correspondingly the turbulence decays. The evolution of the torque and wind kinetic energy is analysed and the period and damping of the oscillations are quantified as a function of Reynolds number.

2:10PM R14.00006 Angular momentum transport and flow super-rotation in Rayleigh stable Taylor-Couette , FREJA NORDSIEK, University of Maryland at College Park, SANDER HUISMAN, ROELAND VAN DER VEEN, CHAO SUN, DETLEF LOHSE, Physics of Fluids group, University of Twente, Netherlands, DANIEL LATHROP, University of Maryland at College Park — We present experimental velocimetry and torque measurements for Taylor-Couette flow in the Rayleigh stable regime. Measurements are taken on two geometrically similar experiments, both of which had axial boundaries attatched to the outer cylinder, which is known to cause Ekman pumping. The Twente experiment has a radius ratio of 0.716, an aspect ratio of 11.68, and measures azimuthal velocities by Laser Doppler Anenometry. The Maryland experiment has a radius ratio of 0.725, an aspect ratio of 11.47, and measures the torque required to rotate the inner cylinder. The torque on the inner cylinder is observed to be greater than that of the analytical Couette profile and has a complex dependence on the Reynolds number and Ω_i/Ω_o. The azimuthal velocity profiles also deviate from the laminar Couette profile. Signficantly, super-rotation in the angular velocity has been observed for $1 > \Omega_i/\Omega_o > 0$. In the quasi-Keplerian regime, the angular momentum profiles consist of an approximately constant inner region connected to an outer region approximately in solid-body rotation at Ω_o, which suggests that angular momentum is being actively transported from the inner region to the axial boundaries.

2:23PM R14.00007 Receptivity Mechanisms in a Rotating Torus: Experiments and Simulations[1] , RICHARD CLARKE, SOPHIE CALABRETTO, SCOTT WALBRAN, JIM DENIER, JOHN CATER, University of Auckland, TRENT MATTNER, University of Adelaide — We consider the flow within a rotating fluid-filled torus subject to a sudden change in angular velocity. Previous DNS computations showed the occurence of boundary-layer separation (Hewitt et al., JFM 688), which was conjectured to be linked with structures observed in the top-down visualisations of Madden & Mullin (JFM 265). These showed a "flow front" in the equatorial plane propagating from the outer wall, the position of which was seen to match well with the separated flow structures seen in the DNS. However, in the experiments a second streak was observed at later times on the opposite wall, not seen in the DNS. To better understand this structure, we present the first measurements of the cross-sectional flow, using PIV on an experiment designed to overcome the optical issues in cross-sectional measurements. These demonstrate both the post-separated flow structures seen in earlier DNS, as well as the appearance of a vortex-pair on the opposite equator. These we believe to be likely candidates for the second fronts noted in the Madden experiments. We hypothesise that this vortex pair is generated by small geometric imperfections, an idea seemingly borne out by striking agreement with new DNS conducted in a modified geometry that better represents experimental reality.

[1]This work is funded by the Royal Society of New Zealand Marsden Fund, and the University of Auckland Doctoral Scholarship Programme

2:36PM R14.00008 Whirling skirts , JAMES HANNA, Virginia Tech, JEMAL GUVEN, Universidad Nacional Autónoma de México, MARTIN MICHAEL MÜLLER, Université de Lorraine — Steady dihedral patterns, consisting of sharply peaked traveling waves, may be observed on a spinning skirt. These qualitative features are captured with a minimal model of flowing material on an inextensible, flexible, generalized-conical sheet rotating about a fixed axis. Analytical results indicate that Coriolis forces are essential for establishing the wave patterns, which arise only for a narrow range of Rossby number.

Tuesday, November 26, 2013 1:05PM - 3:28PM –

Session R16 Aerodynamics V 304 - Karen Mulleners, Leibniz University

1:05PM R16.00001 Unsteady Aerodynamics of Static Airfoils in Reverse Flow , ANDREW LIND, ANYA JONES, University of Maryland — Wind tunnel experiments have been conducted on two-dimensional blunt and sharp trailing edge airfoils held at static angles of attack in reverse flow for three Reynolds numbers. The current work is aimed at advancing the understanding of fully developed reverse flow for high-speed helicopter applications, and evaluates the potential for blunt trailing edge airfoils to mitigate unsteady rotor blade airloads in this flow regime. Time-resolved particle image velocimetry measurements at post-stall angles of attack have revealed the evolution of a trailing edge vortex formed by the roll-up of vorticity generated in a separated shear layer. Proper orthogonal decomposition (POD) was applied to the flow field measurements to improve the identification and tracking of dominant flow structures. Unsteady force balance measurements have captured non-structural vibrations with frequency content which correlates well with that of the temporal coefficients for the first two POD spatial modes. These vibrations vary in frequency with angle of attack and are shown to be linked with trailing edge vortex shedding. The findings presented here give fundamental insight towards the development of efficient rotor blades for high-speed helicopters.

1:18PM R16.00002 Experimental Investigation of Dynamic Stall on an Airfoil with Leading Edge Tubercles[1] , JOHN HRYNUK, DOUGLAS BOHL, Clarkson University — Humpback whales are unique in that their flippers have leading edge "bumps" or tubercles. Past work on airfoils modeled after whale flippers has centered on the static aerodynamic characteristics of these airfoils. In the current work, NACA 0012 airfoils modified with leading edge tubercles are investigated to determine the effect of the tubercles on the dynamic characteristics, specifically on dynamic stall vortex formation, of the airfoils. Molecular Tagging Velocimetry (MTV) is used to measure the flow field around the modified airfoils at nondimensional pitch rates of $\Omega = 0.1$, 0.2, and 0.4. The results show that the characteristics of the dynamics stall vortex are dependent on the location relative to the peak or valley of the leading edge bumps. These characteristics are also found to be different than those observed in dynamic stall on a smooth leading edge airfoil. In specific, the location of the dynamic stall vortex appears to form further aft on the airfoil for the tubercle case versus the smooth case.

[1]This work supported by NSF Grant # 0845882.

1:31PM R16.00003 Flow Structure and Forces on an Airfoil Pitching Asymmetrically at High Reduced Frequency[1] , PATRICK HAMMER, AHMED NAGUIB, MANOOCHEHR KOOCHESFAHANI, Michigan State University — Previous experimental work has shown that non-sinusoidal oscillation of a pitching airfoil can greatly alter the vortical flow structure in the wake. The current study focuses on characterizing the corresponding changes in the resulting force on the airfoil. High-order computations are carried out using the FDL3DI solver developed by Visbal's group at the Air Force Research Laboratory. We will describe the influence of various computational parameters on the ability to capture with high fidelity the vortical flow structure observed experimentally. Results will be presented for the history of lift and drag forces on the airfoil, along the with their mean values, and their connection to the motion history.

[1]This work was supported by AFOSR grant number FA9550-10-1-0342.

1:44PM R16.00004 Influence of the vortex shedding on the time evolution of instantaneous pressure fields and forces in rotating airfoils , ARTURO VILLEGAS, FRANCISCO J. DIEZ, Rutgers University — Time-resolved measurements of instantaneous pressure fields and aerodynamic loads are obtained for rotating airfoils. These allowed evaluating temporal variations in the flow field and were able to capture the evolution of vortex shedding in the wake of the rotating blade. The results show the influence of vortex shedding in the instantaneous loads. These measurements involve obtaining first the velocity field from TR-PIV. This is used to calculate the pressure field from the Poisson pressure equation, and later the forces from the integral momentum equation. The robustness of the measurements is analyzed by calculating the PIV uncertainty, and the independence of the calculated forces. Experimental mean aerodynamic forces are compared to theoretical predictions from the Blade Element Momentum theory (BEM) showing good agreement. The instantaneous pressure varied with time only in the wake due to vortex shedding. This is the first time the evolution of the instantaneous pressure field has been resolved for a rotating airfoil. The contribution to the instantaneous forces from each term in the integral momentum equation is evaluated. The analysis shows that the larger contributions to the normal force coefficient are from the unsteady and the pressure terms while the larger contribution to the tangential force coefficient is from the convective term. The method can be used to measure unsteady forces in rotating airfoils, providing useful information not just for computational studies, but also for aerodynamics, material and structural optimization and safety purposes.

1:57PM R16.00005 Aerodynamics of S809 Airfoil at Low and Transitional Reynolds Numbers , JAIME J. CARRERAS, Universidad del TuraboTexas Tech University, NADER LAAL-DEHGHANI, SERDAR GORUMLU, FARAZ MEHDI, LUCIANO CASTILLO, BURAK AKSAK, JIAN SHENG, Texas Tech University — The S809 is a thick airfoil extensively used in wind turbine design applications and model studies in wind tunnel. With increased interests in reducing energy production cost and understanding turbulence and turbine interactions, scaled down models (Re$\sim 10^3$) are often used as an alternative to full scale field experimentation (Re $> 10^6$). This Reynolds number discrepancy raises the issue of scaling for the airfoil performance from laboratory studies to field scale applications. To the best of our knowledge, there are no studies existing in literature to characterize the lift- and drag-coefficients of S809 airfoil at Re less than 3×10^5. This study is to fill the deficit in the current state of knowledge by performing high resolution force measurements. The lift and drag measurements are carried out in Texas Tech Wind Tunnel Facility using an in-house developed dual-cell force balance. The configuration eliminates the large torque and torsion often accompanied by conventional mounts. This unique design allows us to reach a measurement accuracy of 0.02N (0.1%). Comparative studies are performed on a two-dimensional airfoil with a smooth- as well as a well-engineered surface covered by micro-pillar array to simulate the surface conditions of a real life airfoil.

2:10PM R16.00006 Large eddy simulation with periodic forcing of low-frequency flow oscillation near airfoil stall. , JABER ALMUTAIRI, Assistant Professor, College of Technological Studies, Shuwaikh,70654, IBRAHEEM ALQADI, ELTEYEB ELJACK, Assistant Professor, King Abdulaziz University, Jeddah, Saudi Arabia, 21589 — The effect of periodic forcing on the airfoil performance is investigated in the present study. A large eddy simulation with a flow control technique of periodic forcing is used to remove the low-frequency flow oscillation of the NACA-0012 airfoil at a Reynolds number of 130,000 and incidence of 11.5° and thus enhancing the performance of the airfoil. The periodic forcing is introduced into the laminar boundary layer just upstream of the natural flow separation to produce perturbations in the near wall region. The amplitude of the forcing is set to be 0.3% of the freestream velocity while several different frequencies based on the detected frequency of the vortex trailing edge revealed from the simulation of the natural low-frequency flow oscillation are used. It was found that periodic forcing removes the low-frequency flow oscillation, and as a consequence improve the performance of the airfoil. It has been seen that the large fluctuation of the lift and drag coefficients are entirely eliminated and the enlargement of the turbulent boundary layer that usually occurs after separation and the associated increase of the normal velocity component are reduced sharply when the periodic forcing is added.

2:23PM R16.00007 Analysis of Dynamic Stall Through Chirp Signal Pitch Excursions , KYLE HEINTZ, DUSTIN COLEMAN, MICHAEL WICKS, THOMAS CORKE, FLINT THOMAS, University of Notre Dame — An augmentation of the typical pitching airfoil experiment has been performed where the pitching frequency and amplitude are dynamically varied in a short-time event to produce a "chirp" trajectory, $\alpha(t) = \alpha_0 + \alpha_1(t)\sin(t\omega(t))$. The frequency evolution followed a Schroeder-phase relation, $\omega(t) = \omega_{min} + K(\omega_{max} - \omega_{min})$. The frequencies ranged from 0.5Hz to 30Hz, resulting in reduced frequencies from 0.02 to 0.1. The free-stream Mach number ranged from Mach 0.4 to 0.6, giving chord Reynolds numbers from 5×10^5 to 3×10^6. The airfoil was a NACA 23012 section shape that was fully instrumented with 31 flush-mounted high-bandwidth pressure transducers. The pressure transducer outputs were simultaneously sampled with the instantaneous angle of attack, $\alpha(t)$. The motivation for this study was to compare dynamic stall under non-equilibrium conditions. A particular interest is on the flow features that occur when dynamically passing between light and deep stall regimes. The results include phase analysis of aerodynamic loads, wavelet-based spectral analysis, and the determination of the intra-cycle aerodynamic damping factors.

2:36PM R16.00008 LES of turbulent separated flow over NACA0015 at Reynolds number 1,600,000–toward the separation control by a DBD plasma actuator , MAKOTO SATO, ISAS/JAXA, KENGO ASADA, University of Tokyo, TAKU NONOMURA, SOSHI KAWAI, HIKARU AONO, AIKO YAKENO, KOZO FUJII, ISAS/JAXA — Large eddy simulation of a separated flow over NACA0015 at Reynolds number 1,600,000 with angle of attack 20.1 deg. is conducted to clarify the feature of turbulent separation at high Reynolds number. The grid point is approximately 1 billion, and a high order scheme is used in this simulation. The LES result agrees with experiment data in terms of the laminar-separation bubble region, the locations of reattachment point and second separation point and C_p distribution. In the turbulent separated flow of this simulation, the laminar-separation bubble is formed near the leading edge with turbulent transition, then turbulent boundary layer develops over the airfoil surface and the flow is separated as turbulent separation. Here, streamwise velocities in the attached region correspond to the profile of turbulent boundary layer. In addition, flow structures at Re=1,600,000 are compared to those at Re=63,000 about the turbulent transition, separation behavior, the space scale, time scale and so on. The most unstable frequency of the laminar separation flow at Re=1,600,000 is 10-20 times of that of Re=63,000 The flow scale at transition point of Re=1,600,000 is about 1/15 times of that of Re=63,000.

2:49PM R16.00009 Direct Numerical Simulation of Turbulent Flows over Microstructured Surfaces at $Re_\tau = 180$, JEE HANN NG, RAJEEV KUMAR JAIMAN, TEE TAI LIM, National University of Singapore — Direct numerical simulations (DNSs) are used to investigate drag-reducing performance of microstructured surfaces (i.e. riblets) in a turbulent channel flow. The present numerical study aims to analyze systematically the influence of riblet geometry on the near wall properties and flow structures. In previous studies, a reduction in the skin friction drag was attributed to two mechanisms: (i) riblets shield the wall from the action of near wall streamwise vortices, (ii) mitigate the cross-stream motions of these eddy structures. The present work aims to investigate the validity of these propositions for a various set of riblet configurations. To begin with, a standard V-groove riblet geometry with six different heights and spacings, spanning both drag-reducing and drag-increasing regimes, is considered. These simulations are conducted in a channel flow at a friction Reynolds number $Re_\tau = 180$, where riblets are mounted on one of the walls. The results from these simulations reveal the interactions of the near wall structures with riblets, and the modifications of these flow structures to enable the drag reduction. Finally, the effects of spacing and height of the riblets are summarized for $Re_\tau = 180$.

3:02PM R16.00010 Impact of surface roughness on the turbulent wake flow of a turbine blade , KAREN MULLENERS, Leibniz Universitaet Hannover — Roughened aero engine blade surfaces lead to increased friction and reduced efficiency of the individual blades. The surface roughness also affects the wake flow of the blade and thus the inflow conditions for the subsequent compressor or turbine stage. To investigate the impact of surface roughness on a turbulent blade wake, we conducted velocity field measurements by means of stereo Particle Image Velocimetry (PIV) in the wake of a roughened turbine blade in a linear cascade wind tunnel. The turbine blade was roughened at different chord-wise locations. We examined the influence of the chord-wise location of the added surface roughness by comparing their impact on the width and depth of the wake, the positions and distribution of vortical structures and the overall circulation in the wake. The associated variations in the wake's turbulence characteristics including Reynolds stresses were also explored.

3:15PM R16.00011 Cessna-172R Airplane in Cruise and Landing Configurations: A Numerical Study of the Wing Loads and Wake , PANKAJ JHA[1], The Pennsylvania State University — The present work deals with the analysis of flight test data on a Cessna 172R airplane near University Park airport in Pennsylvania. Several tests pertaining to rate-of-climb, cruise, stall and landing were performed. Those of aerodynamic nature will be discussed. The wing loads for the cruise as well as landing configurations with various flap angles were computed using a vortex method considering horse-shoe and bound vortices. The stall speed and maximum lift coefficient of the airplane for these flap settings at a particular altitude were determined. The comparison against the processed flight data was generally very good. A detailed study will be presented. A CFD approach inspired by the author's work (Jha et al, 2013) to model wind turbine blades and wakes and classical aerodynamics problems was taken to model the airplane wings. The simulation results were also compared against the flight data. In addition, these simulations facilitated visualization and analysis of flow features of interest, like wing tip trailing vortices and their turbulence characterization.

[1]Graduate Research Assistant, Aerospace Engineering

Tuesday, November 26, 2013 1:05PM - 3:15PM –

Session R17 Biofluids: Locomotion XI - Microswimmers and Bacteria II 305 - Mahdi Mohebbi, University of Pittsburgh

1:05PM R17.00001 Microfluidic study of bacterial interactions and attachment to oil drops , GABRIEL JUAREZ, ROMAN STOCKER, MIT — The swimming dynamics of bacteria are strongly influenced by interfaces. Motile bacteria often accumulate near solid-liquid and liquid-liquid interfaces and eventually attach. Attachment of bacteria to these interfaces is crucial for the formation of biofilms (liquid-solid), pellicles (liquid-air), and oil-degrading communities (liquid-liquid). However, the mechanisms of attachment by microbes to these surfaces are not well understood. We have investigated the effect of external flow on the probability of attachment of motile and non-motile marine bacteria to oil droplets in straight microchannels. We find that motility enhances the encounter rate between bacteria and drops leading to an increase in attachment. Characterizing these interactions in the presence of flow will promote an understanding of oil-microbe and particulate-microbe interactions in aquatic environments where the degradation rate of organic matter depends on the attachment and colonization by bacteria.

1:18PM R17.00002 Turbulent flow from a microscopic organism's perspective: What does it feel like to be tiny in the ocean? , RACHEL E. PEPPER, EVAN VARIANO, M.A.R. KOEHL, University of California, Berkeley — Microscopic organisms in the ocean live in turbulent flows. Swimming organisms navigate through the water (e.g. larvae land on substrata, predators find prey), but the mechanisms by which they do so in turbulent flow are poorly understood. Because microscopic organisms are smaller than the Kolmagorov length, they experience turbulence as a Lagrangian time series of varying linear gradients in velocity, and as a Lagrangian time series of varying accelerations. But what are these time series like? While the average gradients, accelerations, and timescales can be estimated from the dissipative scales of the flow, there are indications that organisms are disproportionally affected by intermittent events. Understanding the frequency of such events in different environments is critical to understanding how microorganisms respond to and navigate in turbulent flow. To understand the hydrodynamic cues that microscopic organisms experience in the ocean we measure velocity gradients and accelerations along Lagrangian trajectories in realistic ocean flow on the spatial and temporal scales encountered by such organisms. Here we compare results measured using PIV for hydrodynamic cues above rough biological substrata and smooth substrata, as well as cues near and far from substrata.

1:31PM R17.00003 Squirming At Finite Reynolds Number , NICHOLAS CHISHOLM, ZIYI ZHU, ADITYA KHAIR, Dept. of Chemical Engineering, Carnegie Mellon University — The dynamics of swimming microorganisms at zero Reynolds number ($Re = 0$) has been the subject of extensive theoretical and experimental investigation over the past decade, and the study of locomotion at high Reynolds number ($Re \gg 1$), where inertial forces are dominant, has a venerable history. In this talk, we consider swimming between these limits, i.e. at finite Reynolds numbers, using the popular "squirmer" model of self-propulsion, wherein locomotion is achieved through surface distortions. We first utilize matched asymptotic expansions to derive an analytical expression for the swimming velocity of a squirmer through $O(Re^2)$, which highlights that inertia affects so-called "pusher" and "puller" swimmers in fundamentally different manners. (The equivalent of Whitehead's paradox for a self-propelled object is elucidated in the process.) Next, we employ numerical methods to compute the swimming velocity of pushers and pullers for higher Reynolds numbers. Finally, we demonstrate that inertia causes squirmers (which are non-chiral) to drift across the streamlines of an imposed shear flow.

1:44PM R17.00004 General Squirming Motion in a Stokes Flow , ON SHUN PAK, Dept. of Mechanical and Aerospace Engineering, UC San Diego, ERIC LAUGA, Dept. of Mechanical and Aerospace Engineering, UC San Diego, and Dept. of Applied Mathematics and Theoretical Physics, University of Cambridge — Some microorganisms such as ciliates (*Opalina*) and colonies of flagellates (*Volvox*) are approximately spherical in shape and swim using beating arrays of cilia covering their surfaces. The ciliary motion over the surface may be mathematically modeled as the generation of effectively tangential velocities on the spherical surface – known as squirming motion. Previous analyses assumed axisymmetry and hence restricted all swimming kinematics to take place along a line. Here we remove this limitation and extend the analysis to general non-axisymmetric squirming motion. We derive analytically the three-dimensional translational and rotational swimming velocities as well as the surrounding flow field of a general squirmer. The framework developed here completes the analysis of squirming motion in a Stokes flow.

1:57PM R17.00005 An efficient framework for qualitative and quantitative analysis of magnetically actuated, rigid microswimmers , FARSHAD MESHKATI, University of Nevada, Reno, U KEI CHEANG, MINJUN KIM, Drexel University, HENRY FU, University of Nevada, Reno — Artificial microswimmers or microrobots have been actively investigated for possible applications in microactuation, drug-delivery, in situ sensing and diagnostics, and microtransport and assembly. We describe simple achiral, rigid microswimmers actuated by rotating magnetic fields, and elucidate the the minimal conditions for propulsion. We present an efficient method for analyzing the propulsion of such swimmers, which is capable of predicting the speed and direction of swimming as well as the swimmer's rotational dynamics. The method assumes knowledge of the swimmer's geometry and magnetic dipole moment,which can be measured from its response to an impulsive change in the direction of the magnetic field. The method only requires a single calculation of the swimmer's mobility matrix using a boundary element method such as the method of regularized Stokeslets. We validate our method by finding good agreement with experiment for both qualitative and quantitative predictions. The method described can be easily applied to rigid swimmers with arbitrary geometries which are rotated by external magnetic fields.

2:10PM R17.00006 Copepod Trajectory Characteristics in Thin Layers of Toxic Algal Exudates , D.R. WEBSTER, A.C. TRUE, M.J. WEISSBURG, J. YEN, Georgia Tech — Recently documented thin layers of toxic phytoplankton ("cryptic blooms") are modeled in a custom flume system for copepod behavioral assays. Planar laser-induced fluorescence (LIF) measurements quantify the spatiotemporal structure of the chemical layers ensuring a close match to *in situ* bloom conditions and allowing for quantification of threshold dissolved toxin levels that induce behavioral responses. Assays with the copepods *Acartia tonsa* (hop-sinker) and *Temora longicornis*(cruiser) in thin layers of toxic exudates from the common dinoflagellate *Karenia brevis* (cell equivalent $\sim$ 1 - 10,000 cells/mL) examine the effects of dissolved toxic compounds and copepod species on swimming trajectory characteristics. Computation of parameters such as swimming speed and the fractal dimension of the two-dimensional trajectory (F_{2D}) allows for statistical evaluation of copepod behavioral responses to dissolved toxic compounds associated with harmful algal blooms (HABs). Changes in copepod swimming behavior caused by toxic compounds can significantly influence predator, prey, and mate encounter rates by altering the fracticality ("diffuseness" or "volume-fillingness") of a copepod's trajectory. As trophic mediators linking primary producers and higher trophic levels, copepods can significantly influence HAB dynamics and modulate large scale ecological effects through their behavioral interactions with toxic blooms.

2:23PM R17.00007 Capturing stealthy microswimmers into sphere-bound orbits , DAISUKE TAKAGI, University of Hawaii at Manoa, JEREMIE PALACCI, New York University, ADAM BRAUNSCHWEIG, University of Miami, MICHAEL SHELLEY, JUN ZHANG, New York University — In potential applications ranging from microfluidic mixing to cargo transport, microscopic swimmers must propel themselves through complex environments. However the interaction of swimmers and obstacles is not well understood. Here we study the autonomous movement of catallytically-driven Au-Pt nanorods through a suspension of solid spheres resting on a horizontal plane. Though the spheres exert no net force or torque on the rods, the rods experience a short-range attraction and orbit around the spheres with essentially no decrease in their speed. We propose that the apparent attraction and speed conservation are a consequence of lubrication effects and the phoretic propulsion mechanism of nanorods. This suggests strategies to capture various self-propelled bodies and motile cells in confined spaces.

2:36PM R17.00008 ABSTRACT WITHDRAWN –

2:49PM R17.00009 The sperm flagellum counterbend phenomenon , HERMES GADELHA, EAMONN GAFFNEY, ALAIN GORIELY, University of Oxford — Recent observations of flagellar counterbend in sperm show that the mechanical induction of curvature in one part of a passive flagellum induces a compensatory countercurvature elsewhere. This apparent paradoxical effect cannot be explained using the standard elastic rod theory of Euler and Bernoulli, or even the more general Cosserat theory of rods. Here, we develop a mechanical model capable of predicting the curvature reversal events observed in eukaryotic flagella. This is achieved by allowing the interaction of deformations in different material directions, by not only accounting for structural bending, but also the elastic forces originating from the cross-linking mechanics. Large amplitude configurations can be described analytically and an excellent match between the model and the observed counterbend deformation was found. This allowed a simultaneous estimation of multiple sperm flagellum material parameters, namely the cross-linking sliding resistance, the bending stiffness and the sperm head junction compliance ratio. Our analysis demonstrates that the counterbend emerges as a fundamental property of sliding resistance, which also suggests that cross-linking proteins may contribute to the regulation of the flagellar waveform in swimming sperm via counterbend mechanics. Finally, we investigate how the counterbend-type dynamics in sperm flagella is affected by viscous dissipation.

3:02PM R17.00010 Stability of a downflowing gyrotactic microorganism suspension: formation of blips , YONGYUN HWANG, TIM PEDLEY, DAMTP, University of Cambridge — Hydrodynamic focusing of cells is a robust feature in downflowing suspensions of swimming gyrotatic microorganisms. In the early experiments with a downward pipe flow, Kessler (1986, *J. Fluid Mech*, 173:191-205) observed that the focussed beam-like structure of cells in the region of most rapid downflow exhibits regular-spaced axisymmetric blips, but the mechanism by which the blips are formed has not been well understood yet. For this purpose, we perform a linear stability analysis of a downflowing suspension of randomly swimming gyrotactic cells in a two-dimensional vertical channel. For relatively small flow rates, the focussed beam in the channel exhibits a varicose instability strikingly similar to the blips in the pipe flow, and this becomes gradually damped out as the flow rate increases. It is found that the varicose instability essentially originates from the interaction of cell-concentration fluctuations with the horizontal gradient of the cell-orientation vector field, which does not appear in unform suspensions. A comparison is finally made with recent experimental results by Croze & Bees (2013, In preparation), showing qualitatively good agreement.

Tuesday, November 26, 2013 1:05PM - 3:28PM –

Session R18 CFD IX 306/307 - Mehdi B. Nik, University of Pittsburgh

1:05PM R18.00001 Nonlinear optimisation of scalar mixing in plane Poiseuille flow with finite diffusivity , C.P. CAULFIELD, BP Institute & DAMTP, University of Cambridge, DIMITRY FOURES, DAMTP, University of Cambridge, P.J. SCHMID, LadHyX, CNRS-Ecole Polytechnique — We consider the nonlinear optimisation of the mixing of a passive scalar, initially arranged in two layers, in 2D plane Poiseuille flow at finite Reynolds number and Péclet number, $Re \sim Pe \sim O(10^3)$. We use a nonlinear-adjoint-looping approach to minimise the variance of the scalar concentration θ at various target times T, subject either to a finite kinetic energy initial disturbance, or wall velocity perturbation. We show that both optimal initial perturbations and optimal wall excitation strategies which minimise the variance of θ are distinct from the equivalent perturbations which maximise the time-averaged energy gain of disturbance at $t = T$, and that these "gain" perturbations can often be poor at scalar mixing. We also identify perturbations and excitation strategies which minimise the distribution of θ at the target time relative to a particular Sobolev norm of negative index, a "mix-norm" as used in flows with no diffusion to measure "mixing" in the sense of ergodic theory (G. Mathew, I. Mezic, & L. Petzold 2005 *Physica D*, **211**, 23-46). We show the close connection between these mix-norm perturbations and the optimal variance perturbations, all of which initially increase gradients to ensure good mixing at later times.

1:18PM R18.00002 Simulation-based optimization using finite-time approximations of the infinite-time-average statistics , POORIYA BEYHAGHI, THOMAS BEWLEY, University of California, San Diego — Simulated-based optimization problems are difficult, due both to the nonconvexity of the cost function and to the extreme cost of accurate function evaluations, especially if the cost function is derived from a finite-time-average approximation of the infinite-time-average statistics. In this work, we have developed an algorithm that controls both the location and the accuracy of each cost function evaluation. At each step of the algorithm, a Delaunay triangulation is created based on the existing evaluation points. In each simplex so created, the algorithm optimizes a cost function based on a polyharmonic spline regression. At each optimization step, an appropriately-modeled error function is combined with the regression, weighted with a tuning parameter governing the trade-off between local refinement and global exploration. In this way, the location of the new candidate point for the global minimum is found; then, based on the value of the minimum available cost function evaluations and the uncertainty associated with it, an efficient finite-time approximation at this point is calculated. The global convergence of this algorithm will be shown, and its efficiency will be tested on representatives test functions.

1:31PM R18.00003 Determining wave resistance of a ship using a dissipative potential flow model[1] , MIRJAM FÜRTH, MINGYI TAN, ZHI-MIN CHEN, University of Southampton, Fluid Structure Interactions Research Group — Potential flow modelling is a common method to predict the wave resistance of ships. In its conventional form the flow is assumed to be free from damping due to the inviscid assumption of potential flow. However, it is evident by just looking at waves that they decay with time and distance. It is a reasonable assumption that, by including more of the actual physical aspect in mathematical model, the quality of the prediction will improve. As Havelock wrote almost 80 years ago "It seems fairly certain that one of the main causes of differences between theoretical and experimental result is the neglect of fluid friction in the calculation of ship waves." In this study, the problem is modelled using Kelvin sources with a translating speed. Rayleigh damping is introduced in the model to emulate viscous damping. To calculate the source influences, a dissipative 3D Green function is derived. For initial validation of the Green function, thin ship theory is used to determine the wave pattern behind a Wigley hull and a modified form of the Eggers et al. transverse cut technique is used to calculate the wave resistance. To evaluate the method for fuller and more realistic hull shapes a panel method which calculates the resistance via the pressure on the ship hull is used.

[1]This project is sponsored by Lloyds Register Foundation, their support is greatly appreciated.

1:44PM R18.00004 Thermal Fluctuations in Smooth Dissipative Particle Dynamics simulation of mesoscopic thermal systems , NIKOLAOS GATSONIS, JUN YANG, Worcester Polytechnic Institute — The SDPD-DV is implemented in our work for arbitrary 3D wall bounded geometries. The particle position and momentum equations are integrated with a velocity-Verlet algorithm and the entropy equation is integrated with a Runge-Kutta algorithm. Simulations of nitrogen gas are performed to evaluate the effects of timestep and particle scale on temperature, self-diffusion coefficient and shear viscosity. The hydrodynamic fluctuations in temperature, density, pressure and velocity from the SDPD-DV simulations are evaluated and compared with theoretical predictions. Steady planar thermal Couette flows are simulated and compared with analytical solutions. Simulations cover the hydrodynamic and mesocopic regime and show thermal fluctuations and their dependence on particle size.

1:57PM R18.00005 Boundary conditions for coupling molecular dynamics simulations to continuum simulations[1] , LIV HERDMAN, YVES DUBIEF, University of Vermont — Coupling continuum simulations to molecular dynamics simulations require implementing boundary conditions that constrain the atomic motions to match the physical properties of the large-scale simulations. The traditional wall and periodic boundary conditions used in molecular dynamics present difficulties for simulating non-equilibrium and spatially evolving flows. We are working toward creating an evolving boundary condition to match temperature and momentum conditions in atomistic simulations that are driven by coupled continuum simulations. We have developed an inlet boundary condition that utilizes a periodic buffer cell to drive the variable thermodynamic and flow conditions. In this work we demonstrate the new inlet boundary condition with simulations of a Lenard-Jones fluid in a channel and compare the effects of different outlet boundary conditions

[1]This work is support by NASA NNX11AM07A

2:10PM R18.00006 Patient Specific Multiscale Simulations of Blood Flow in Coronary Artery Bypass Surgery , ABHAY BANGALORE RAMACHANDRA, SETHURAMAN SANKARAN, Department of Mechanical and Aerospace Engineering, UCSD, ANDREW M. KAHN, Department of Medicine, UCSD, ALISON L. MARSDEN, Department of Mechanical and Aerospace Engineering, UCSD — Coronary artery bypass surgery is performed to revascularize blocked coronary arteries in roughly 400,000 patients per year in the US.While arterial grafts offer superior patency, vein grafts are used in more than 70% of procedures, as most patients require multiple grafts. Vein graft failure (approx. 50% within 10 years) remains a major clinical issue. Mounting evidence suggests that hemodynamics plays a key role as a mechano-biological stimulus contributing to graft failure. However, quantifying relevant hemodynamic quantities (e.g. wall shear stress) invivo is not possible directly using clinical imaging techniques. We numerically compute graft hemodynamics in a cohort of 3-D patient specific models using a stabilized finite element method. The 3D flow domain is coupled to a 0D lumped parameter circulatory model. Boundary conditions are tuned to match patient specific blood pressures, stroke volumes & heart rates. Results reproduce clinically observed coronary flow waveforms. We quantify differences in multiple hemodynamic quantities between arterial &venous grafts & discuss possible correlations between graft hemodynamics & clinically observed graft failure.Such correlations will provide further insight into mechanisms of graft failure and may lead to improved clinical outcomes.

2:23PM R18.00007 Coupling surface and subsurface flows with curved interfaces , PU SONG, IVAN YOTOV, Department of Mathematics, University of Pittsburgh, 301 Thackeray, Pittsburgh, Pennsylvania 15260, USA — A mortar multiscale method is developed for the coupled Stokes andDarcy flows with the Beavers–Joseph–Saffman interface condition in irregular domains. Conforming Stokes elements and multipoint flux mixed finite elements in Darcy are used to discretize the subdomains on the fine scale. A coarse scale mortar finite element space is used to approximate interface stresses and pressures and impose weakly continuity of velocities and fluxes. Matching conditions on curved interfaces are imposed by mapping the physical grids to reference grids with flat interfaces.

2:36PM R18.00008 An efficient pressure-correction method for incompressible multifluid flows[1] , M. DODD, A. FERRANTE, University of Washington, Seattle — We present a new pressure-correction (PC) method for solving incompressible multifluid flows with large density ratios. The novelty of the method is that the variable coefficient Poisson equation that arises in solving the variable-density Navier-Stokes equations has been reduced to a constant coefficient equation, which can then be solved directly using a fast Poisson solver. The new method is coupled to our mass-conserving volume-of-fluid (VoF) method to capture the interface between the moving fluids. First, we verified the new PC/VoF solver using the capillary wave test-case up to density and viscosity ratios of 10,000. Then, we validated the new flow solver by simulating the motion of a falling water droplet in air by comparing the droplet terminal velocity with the experimental value (Beard, 1976) for $95.6 \leq \mathrm{Re} \leq 473$, $0.06 \leq \mathrm{We} \leq 0.61$, and $0.05 \leq \mathrm{Bo} \leq 0.26$. We also verified the solver for a rising air bubble in water. The algorithm is shown to be second-order accurate, and stable for density and viscosity ratios up to 10,000. Also, we show that our fast Poisson solver is more than ten times faster than the Hypre multigrid solver up to a 1024^3 grid and 1024 cores.

[1]NSF CAREER #1054591

2:49PM R18.00009 A Controls-CFD Approach for Estimation of Concentration from a Moving Aerial Source: Advantages of a Finite Volume-TVD implementation with Guidance-Based Grid Adaptation , TATIANA EGOROVA, NIKOLAOS A. GATSONIS, MICHAEL A. DEMETRIOU, Worcester Polytechnic Institute — In this work the process of gas release into the atmosphere by a moving aerial source is simulated and estimated using a sensing aerial vehicle (SAV). The process is modeled with atmospheric advection diffusion equation, which is solved by the finite volume method (FVM). Advective fluxes are constrained using total variation diminishing (TVD) approach. The estimator provides on-line estimates of concentration field and proximity of the source. The guidance of the SAV is dictated by the performance of the estimator. To further improve the estimation algorithm from the computational prospective, the grid is adapted dynamically through local refinement and coarsening. The adaptation algorithm uses the current sensor position as a center of refinement, with the areas further away from the SAV being covered by a coarse grid. This leads to the time varying state matrix of the estimator and the variation depends on the SAV motion. Advantages of the adaptive FVM-TVD implementation are illustrated on the examples of estimator performance for different source trajectories.

3:02PM R18.00010 Interface capturing using a compressive advection method and a compositional modelling approach: Applications[1] , DIMITRIOS PAVLIDIS, ZHIHUA XIE, JAMES PERCIVAL, JEFFERSON GOMES, CHRISTOPHER PAIN, OMAR MATAR, Imperial College London — Progress on a consistent approach for interface-capturing in which each component represents a different phase/fluid is described. The aim is to develop a general multiphase modelling approach based on fully-unstructured meshes that can exploit the latest mesh adaptivity methods, and in which each fluid phase may have a number of components. The method is based on the P1DG-P2 finite element pair, in which the velocity has a linear discontinuous variation and the pressure has a quadratic continuous variation. The method is compared against experimental results for a collapsing water column test case and a convergence study is performed. The method is then used to simulate horizontal slug flow.

[1]EPSRC Programme Grant EP/K003976/1.

3:15PM R18.00011 Diffused interface ghost fluid method for incompressible multiphase, phase change simulations , MOON SOO LEE, AMIR RIAZ, University of Maryland — Sharp interface methods for simulating multiphase flow often suffer from unstable pressure and velocity fluctuations for problems involving mass transfer. An improved sharp interface method is developed for multiphase flow with phase change using both sharp and diffused interfacial properties. The approach is based on defining continuous, phase averaged velocity and density fields within a diffused interfacial region while using the sharp treatment for the implementation of the jumps in the pressure and the temperature gradient. The method implements interface advection with diffused and stable velocity field but can represent accurate movement of the sharp interface. Two-dimensional film boiling problems are solved on a horizontal surface to demonstrate the performance of the new approach.

Tuesday, November 26, 2013 1:05PM - 3:15PM –

Session R19 Boundary Layers VIII: Experimental 310/311 - Ellen Longmire, University of Minnesota

1:05PM R19.00001 Large field of view multi-resolution experimental measurement of the spatial structure of a high Reynolds number turbulent boundary layer , NICOLAS A BUCHMANN, CALLUM ATKINSON, Monash University, CHARITHA DE SILVA, EBENEZER GNANAMANICKAM, NICHOLAS HUTCHINS, Melbourne University, JULIO SORIA, Monash University, IVAN MARUSIC, Melbourne University — A multiple camera, multiple resolution PIV system consisting of nine highe resolution cameras and two dual cavity lasers is used to instantaneously measure a large streamwise, wall-normal field of view ($2\delta \times 1.5\delta$), while simultaneously resolving the near-wall flow down to a height of y $+ \approx$ 3 at Reynolds numbers of Re$\tau \approx$ 8, 000 to 30, 000. The high spatial resolution in the near-wall vicinity enables direct estimation of the wall shear stress and allows for subsequent and conditional analysis of both the near-wall and larger outer flow structures and fluctuations. The quality of this measurement is assessed via statistical comparison with previous measurements made using hot-wire anemometry and compared with the highest available Reynolds number DNS. Conditional two-point correlations, and conditional mean structures will be presented.

1:18PM R19.00002 Turbulence measurements in high Reynolds number boundary layers[1] , MARGIT VALLIKIVI, Princeton University, ALEXANDER SMITS, Princeton University and Monash University — Measurements are conducted in zero pressure gradient turbulent boundary layers for Reynolds numbers from Re$_\theta$ = 9,000 to 225,000. The experiments were performed in the High Reynolds number Test Facility (HRTF) at Princeton University, which uses compressed air as the working fluid. Nano-Scale Thermal Anemometry Probes (NSTAPs) are used to acquire data with very high spatial and temporal precision. These new data are used to study the scaling behavior of the streamwise velocity fluctuations in the boundary layer and make comparisons with the scaling of other wall-bounded turbulent flows.

[1]Supported under ONR Grant N00014-09-1-0263 (program manager Ron Joslin) and NSF Grant CBET-1064257 (program manager Henning Winter).

1:31PM R19.00003 Effect of pressure gradient fluctuations on boundary layer turbulence[1] , PRANAV JOSHI, JOSEPH KATZ, Johns Hopkins University, XIAOFENG LIU, Baltimore Aircoil Company — The present study focuses on the effect of large-scale pressure gradient fluctuations on turbulence in both, zero pressure gradient (ZPG) and mean favorable pressure gradient (FPG) boundary layers. Time-resolved, two-dimensional PIV data in the streamwise-wall-normal plane enables us to calculate the instantaneous pressure distributions by integrating the planar projection of the material acceleration of the fluid. In both boundary layers, sweeps ($u' > 0$, $v' < 0$) mostly occur during periods of adverse pressure gradient fluctuations ($\partial p'/\partial x > 0$), while favorable pressure gradient fluctuations ($\partial p'/\partial x < 0$) accompany ejections ($u' < 0$, $v' > 0$). Conditional averaging indicates that in the ZPG boundary layer, large-scale $\partial p'/\partial x > 0$ events accompanying sweeps lead to the formation of a growing region of ejection downstream, in a phenomenon resembling adverse-pressure induced flow separation. This phenomenon is much less pronounced in the FPG boundary layer, as the large-scale $\partial p'/\partial x > 0$ events are for the most part significantly weaker than the mean FPG. Conditional sampling and instantaneous data in the ZPG boundary layer also confirm that although some of the ejections are preceded, and presumably initiated, by regions of adverse pressure gradients and sweeps, others are not. In the FPG boundary layer, there is no evidence of sweeps or adverse pressure gradients immediately upstream of ejections. The mechanisms initiating these structures presumably occur far upstream of the peak in favorable pressure gradient fluctuations.

[1]Sponsored by NSF, CBET Division, Fluid Dynamics program.

1:44PM R19.00004 Boundary layer response to periodic spanwise perturbation by an array of obstacles , YAN MING TAN, ELLEN LONGMIRE, University of Minnesota — Hot-wire and stereoscopic particle image velocimetry (SPIV) measurements were performed on zero pressure gradient turbulent boundary layers perturbed by a single array of cylinders extending into the log region. The array spacing had a strong effect on the resulting vortex packet organization. With 0.6δ array spacing, incoming vortex packets are frequently enhanced and remain stable over a downstream distance of 7δ. On the other hand, a 0.2δ spacing strongly disturbed incoming packets initially, but the incoming organization frequently reappeared by 2δ downstream. Measurements indicate that the effect of the perturbation is strongest in the mid-span location between array elements. Furthermore, hot-wire measurements suggest that the strongest velocity perturbations migrate upwards with increasing downstream distance. SPIV results in streamwise-spanwise planes at different heights will be compared to show how structural aspects of the perturbation vary with downstream distance.

1:57PM R19.00005 Latest Developments on Obtaining Accurate Measurements with Pitot Tubes in ZPG Turbulent Boundary Layers , HASSAN NAGIB, RICARDO VINUESA, IIT, Chicago — Ability of available Pitot tube corrections to provide accurate mean velocity profiles in ZPG boundary layers is re-examined following the recent work by Bailey *et al.* Measurements by Bailey *et al.*, carried out with probes of diameters ranging from 0.2 to 1.89 mm, together with new data taken with larger diameters up to 12.82 mm, show deviations with respect to available high-quality datasets and hot-wire measurements in the same Reynolds number range. These deviations are significant in the buffer region around $y^+ = 30 - 40$, and lead to disagreement in the von Kármán coefficient κ extracted from profiles. New forms for shear, near-wall and turbulence corrections are proposed, highlighting the importance of the latest one. Improved agreement in mean velocity profiles is obtained with new forms, where shear and near-wall corrections contribute with around 85%, and remaining 15% of the total correction comes from turbulence correction. Finally, available algorithms to correct wall position in profile measurements of wall-bounded flows are tested, using as benchmark the corrected Pitot measurements with artificially simulated probe shifts and blockage effects. We develop a new scheme, $\kappa B-$Musker, which is able to accurately locate wall position.

2:10PM R19.00006 ABSTRACT WITHDRAWN –

2:23PM R19.00007 Direct measurement of the velocity joint probability density function and higher order moments in turbulent boundary layer flows[1] , JULIO SORIA, CALLUM ATKINSON, Monash University — This work shows how the joint probability density function (JPDF) of the streamwise and wall normal velocity components of a zero-pressure gradient turbulent boundary layer (ZPG-TBL) can be directly measured using the methodology and theoretical framework proposed by Soria & Willert (2012) MST 23, 065301. Higher order moments including Reynolds stresses can be computed directly from two-component (2C) JPDFs of the streamwise and wall normal velocity components by taking moments of the 2C-JPDF. The base data for the direct measurement of the 2C-JPDF are single-exposed image pairs typically used to determine instantaneous 2C-2D particle image velocimetry (PIV) fields. However, in the new direct measurement method, the instantaneous velocity samples necessary to build up the JPDF never need to be determined, which avoids the problems in PIV due to large velocity gradients that are typically encountered in turbulent wall-bounded flows. This new method has been applied to single-exposed image pairs acquired over a range of Reynolds numbers ranging up to $Re_\tau = 19500$ in ZPG-TBL experiments. This paper presents directly measured 2C-JPDFs across the ZPG-TBL as well as higher moment distributions determined from these 2C-JPDFs.

[1]The financial support of the Australian Research Council to undertake this research is gratefully acknowledged.

2:36PM R19.00008 Measurement of entropy generation within bypass transitional flow[1] , RICHARD SKIFTON[2], RALPH BUDWIG[3], DONALD MCELIGOT, JOHN CREPEAU, University of Idaho — A flat plate made from quartz was submersed in the Idaho National Laboratory's Matched Index of Refraction flow facility. Particle Image Velocimetry and Particle Tracking Velocimetry were utilized to capture spatial vector maps at near wall locations *within a transitional boundary layer*. Entropy generation was then calculated directly from measured velocity vector fields using an integral approach. Two flows were studied: a Zero Pressure Gradient (ZPG) and an Adverse Pressure Gradient (APG), with $\beta \approx$ -0.039. Near the leading edge of the plate, the free stream turbulence intensity (FSTI) to drive bypass transition was 7.5% and 4.25% for the ZPG and APG, respectively. Towards the downstream side of the plate, the FSTI was 2.5% and 3% for ZPG and APG, respectively. The integral approach for entropy generation rate, within the transitional region of flow, will be utilized as a design parameter to systematically reduce losses. As a second observation, the entropy generation can be shown to predict the onset of transitional flow.

[1]This research was funded by the DOE EPSCoR program No. DE-SC0004751, and by funding received from the DOE Office of Nuclear Energy's Nuclear Energy University Programs [2]Center for Advanced Energy Studies [3]Center for Ecohydraulics Research

2:49PM R19.00009 Logarithmic Boundary Layers in Strong Taylor-Couette Turbulence , DETLEF LOHSE, SANDER HUISMAN, University of Twente, RODOLFO OSTILLA, Univ of Twente, SVEN SCHARNOWSKI, CHRISTIAN CIERPKA, CHRISTIAN KÄHLER, Univ. Bundeswehr Muenchen, ROBERTO VERZICCO, Univ of Rome Tor Vergata, CHAO SUN, Univ of Twente, SIEGFRIED GROSSMANN, Univ of Marburg — We provide direct measurements of boundary layer profiles in highly turbulent Taylor-Couette flow up to $Re = 2 \times 10^6$ using high-resolution particle image velocimetry and particle tracking velocimetry, complemented by DNS data on the same system up to $Re = 10^5$. We find that the mean azimuthal velocity profile at the inner and outer cylinder can be fitted by the von Kármán log law, but with corrections due to the curvature of the cylinder, which we theoretically account for, based on the Navier-Stokes equation and a closure assumption for the turbulent diffusivity. In particular, we study how these corrections depend on the cylinder radius ratio and show that they are different for the boundary layers at the inner and at the outer cylinder.

3:02PM R19.00010 Logarithmic region of turbulent boundary layer from low to high Reynolds numbers , YUKI WADA, YOSHIYUKI TSUJI, Nagoya University — Recent high Reynolds number experiments indicate that von Karman constant (κ) is close to 0.38. In the classical moderate Reynolds number experiments, however, show the value of κ =0.41. In this study we would like to discuss the discrepancy between these two values. Analyzing the data in the range of 1500<Re_θ<27300, we suggest the idea of Quasi Logarithmic Region (QLR) in which the log law is approximately satisfied and discuss the above issue. Using the QLR, we report the Reynolds number dependence of κ, and recommend the appropriate log-law region.

Tuesday, November 26, 2013 1:05PM - 3:15PM –

Session R20 Boundary Layers IX: Numerical Simulation 315 - NR Panchapakesan, Indian Institute of Technology Madras

1:05PM R20.00001 Large eddy simulation study of the logarithmic law for high-order moments in turbulent boundary layers[1] , RICHARD STEVENS, MICHAEL WILCZEK, CHARLES MENEVEAU, Johns Hopkins University — Analyzing highly resolved experimental data, recently generalized log-laws have been observed by Meneveau and Marusic (JFM 719, R1 (2013)) for higher-order even moments of the streamwise velocity fluctuations u'. These observations provide new and robust data in canonical boundary layers that can be compared to the output of Large Eddy Simulation models. To utilize the new data for this purpose, we have performed large eddy simulations at various resolutions and with different sub-grid models (Smagorinsky and scale-dependent Lagrangian dynamic model). In agreement with the experimental data, the higher-order generalizations of the log-law are also observed in the LES results. But since large eddy simulations do not resolve the sub-grid contributions, we furthermore discuss the possibility of an a-posteriori correction of the statistical moments based on quasi-Gaussian assumptions for the statistics of the sub-grid contributions. We find that, while the corrections are small in the bulk of the flow, they can reach significant amplitudes in the near-wall region.

[1] "Fellowships for Young Energy Scientists" (YES!) of FOM, DFG funding WI 3544/2-1, and US National Science Foundation grants numbers CBET 1133800 and OISE 1243482.

1:18PM R20.00002 Inertial Subrange Spectra in the Log-Law Layer of Turbulent Channel Flow , YUKIO KANEDA, Aichi Institute of Technology, KOJI MORISHITA, Kobe University, TAKASHI ISHIHARA, Nagoya University — High resolution direct numerical simulations (DNSs) of turbulent channel flows with the friction Reynolds number Re_τ up to 5120 show that there exists a layer at y^+ being approximately between 200 and 1200, in which the mean velocity profile and the diagonal components of the inertial subrange velocity correlation spectra fit well to the logarithmic law and the $k^{-5/3}$ law, respectively. Here y^+ is the distance from the wall normalized by the wall unit, and k is the wavenumber in the stream wise direction. The DNS data suggest that in the layer (log-law layer), there exists a high wave number range in which the influence of the mean flow on the turbulence statistics may be regarded to be small as compared to that of nonlinear interactions between the small-scale eddies of size $\sim 1/$(wave number), so that the former influence may be treated as a perturbation added to the turbulent state determined by the nonlinear turbulence dynamics in the absence of the mean flow. A perturbation analysis on the basis of this idea yields a simple prediction for the anisotropic velocity correlation spectra in the inertial subrange. The DNS data agree fairly well with the prediction.

1:31PM R20.00003 Numerical experiments of thermal convection with shear , CURTIS HAMMAN, PARVIZ MOIN, Center for Turbulence Research, Stanford University — Inspired by Sayadi, Hamman and Moin's (2013, J. Fluid Mech.) finding that late-stage boundary layer transition shares the same structure and scaling behaviour of high Reynolds number wall turbulence, we explore whether facets of the related ultimate state of Rayleigh-Bénard convection (RBC) can develop at moderate Rayleigh numbers ($10^5 < Ra < 10^{10}$) with amplified wall shear. In shear-free RBC, turbulence in the bulk produces a large-scale circulation or mean wind too weak to prompt boundary layer transition by shear instability, except possibly at extreme Rayleigh numbers where the largest eddies organize the local wall shear flow above a critical friction Reynolds number. We propose a numerical experiment to produce turbulent, three-dimensional, thermal and kinetic boundary layers near the walls while maintaining shear-free, buoyant turbulence production in the bulk as in unperturbed RBC. We speculate that this flow structure may correlate with the ultimate state of thermal convection, given that most of the near-wall turbulent energy production by shear is due to the effects of a mean wind-induced turbulent wall layer. Simulation results are presented to test this connection between the boundary layer structure of RBC and canonical wall-bounded turbulent shear flows.

1:44PM R20.00004 LES of spatially developing turbulent boundary layer over a concave surface , SUNIL AROLLA, Cornell University, PAUL DURBIN, Iowa State University — We revisit the problem of spatially developing turbulent boundary layer over a concave surface. Unlike previous investigations, we simulate the combined effects of curvature-induced pressure gradients as well as streamline curvature on the turbulence. Our focus is on investigating the response of the turbulent boundary layer to the sudden onset of curvature and the destabilizing influence of concave surface in the presence of pressure gradients. This is of interest for evaluating the turbulence closure models. Numerical simulations have been performed using the large eddy simulation framework in OpenFOAM. The dynamic Smagorinsky model is used to account for the sub-grid scale stresses. A variant of the recycling and rescaling method is used to generate the inflow turbulence. At the beginning of the curve, the momentum thickness Reynolds number is 1300 and the ratio of boundary layer thickness to the radius of curvature is $\delta_0/R = 0.055$. The radial profiles of the mean velocity and turbulence statistics at different locations along the concave surface are presented. In addition, the secondary flow structures observed are reported.

1:57PM R20.00005 The high-order statistics of APG turbulent boundary layers[1] , YVAN MACIEL, Laval University, AYSE G. GUNGOR, Istanbul Technical University, MARK P. SIMENS, U. Politécnica Madrid, JULIO SORIA, Monash University — One and two-point statistics are presented from a new direct numerical simulation of an adverse pressure gradient boundary layer, at $Re_\theta = 250 - 2175$, in which the transition to turbulence is triggered by a trip wire which is modeled using the immersed boundary method. Mean velocity results in the attached turbulent region do not show log law profiles. Departure from the law of the wall occurs throughout the inner region. The production and Reynolds stress peaks move to roughly the middle of the boundary layer. The profiles of the uv correlation factor reveal that de-correlation between u and v takes place throughout the boundary layer, but especially near the wall, as the mean velocity defect increases. The non-dimensional stress ratios and quadrant analysis of uv indicate changes to the turbulence structure. The structure parameter is low, similar to equilibrium APG flows and mixing layers in the present flow and seems to be decreasing as the mean velocity defect increases. The statistics of the upper half of the APG flow show resemblance with results for a mixing layer.

[1]Funded in part by ITU, NSERC of Canada, ARC Discovery Grant, and Multiflow program of the ERC.

2:10PM R20.00006 The structure of APG turbulent boundary layers[1] , AYSE G. GUNGOR, Istanbul Technical University, YVAN MACIEL, Laval University, MARK P. SIMENS, U. Politécnica Madrid, JULIO SORIA, Monash University — A boundary layer under influence of a strong APG is studied using DNS. Transition to turbulence is triggered using a trip wire which is modelled using the immersed boundary method. The Reynolds number close to the exit of the numerical domain is $Re_\theta = 2175$ and the shape-factor $H = 2.5$. Two dimensional two-point spatial correlation functions are obtained in this region and close to the transition region. C_{vu} with a reference point close to the transition region shows a flow periodicity until $Re_\theta \approx 1600$. This periodicity is related to the shear layer instability of the separation bubble created as a result of the APG. The C_{vv} and C_{ww} correlations obtained far from the transition region at $Re_\theta = 2175$ and at $y/\delta = 0.4$ coincide with results obtained for a ZPG boundary layer. Implying that the structure of the v, w fluctuations is the same as in ZPG. However, C_{uu} indicates that the structure of the u fluctuation in an APG boundary layer is almost twice as short as the ZPG structures. The APG structures are also less correlated with the flow at the wall. The near wall structure of strong APG flows is different from ZPG flows in that streaks are much shorter or absent.

[1]Funded in part by ITU, NSERC of Canada, ARC Discovery Grant, and Multiflow program of the ERC.

2:23PM R20.00007 Large eddy simulation of zero-pressure-gradient turbulent boundary layer based on different scaling laws[1] , WAN CHENG, RAVI SAMTANEY, King Abdullah University of Science and Technology — We present results of large eddy simulation (LES) for a smooth-wall, zero-pressure-gradient turbulent boundary layer. We employ the stretched vortex sub-grid-scale model in the simulations augmented by a wall model. Our wall model is based on the virtual-wall model introduced by Chung & Pullin (J. Fluid Mech 2009). An essential component of their wall model is an ODE governing the local wall-normal velocity gradient obtained using inner-scaling ansatz. We test two variants of the wall model based on different similarity laws: one is based on a log-law and the other on a power-law. The specific form of the power law scaling utilized is that proposed by George & Castillo (Appl. Mech. Rev. 1997), dubbed the "GC Law". Turbulent inflow conditions are generated by a recycling method, and applying scaling laws corresponding to the two variants of the wall model, and a uniform way to determine the inlet friction velocity. For Reynolds number based on momentum thickness, Re_θ, ranging from 10^4 to 10^{12} it is found that the velocity profiles generally follow the log law form rather than the power law. For large Reynolds number asymptotic behavior, LES based on different scaling laws the boundary layer thickness and turbulent intensities do not show much difference.

[1]Supported by a KAUST funded project on large eddy simulation of turbulent flows. The IBM Blue Gene P Shaheen at KAUST was utilized for the simulations.

2:36PM R20.00008 Turbulent Boundary Layers in Absence of Mean Shear , BLAIR JOHNSON, EDWIN COWEN, Cornell University — Environmental flows are often observed in which turbulence levels significantly exceed what would be expected from mean boundary shear (e.g. breaking surface waves). This enhanced turbulence produces sediment resuspension and boundary layers that differ greatly from classic turbulent boundary layer characterizations. To identify the contribution of turbulence to such sediment resuspension, experiments are conducted in a facility designed to generate homogeneous isotropic turbulence in absence of mean shear via a Randomly Actuated Synthetic Jet Array (RASJA). Using particle image velocimetry (PIV), boundary layers above both a solid glass bed and a narrowly graded sediment bed are characterized by their mean flows, turbulent kinetic energy, dissipation, spectra, and Reynolds stress. Furthermore, a surprising observation includes the formation of ripple patterns when the turbulence decays above the sediment bed. We hypothesize that the ripples scale with the integral length scale of the turbulence. By varying the percentage of active jets and the relative on- and off-times of jets in the RASJA, our investigations consider the impact of altering the integral length scale of the facility on the resulting turbulent structures and sediment motions observed.

2:49PM R20.00009 Investigation of Turbulent Wedge Spreading Mechanism and How to Reduce Spreading Using Surface Textures[1] , JEFF CHU, DAVID GOLDSTEIN, University of Texas at Austin, GARRY BROWN, Princeton University — We investigate the physics of turbulent wedge spreading in a nominally zero pressure gradient laminar boundary layer over a flat wall using incompressible spectral DNS and an immersed boundary method. Turbulent wedges are simulated over both physical and unphysical surfaces to identify the important factors leading to wedge spreading and turbulence regeneration. Vortex mechanics are examined in detail to elucidate the details of vorticity generation. We find that turbulent wedge spreading appears to rely on tilting spanwise vorticity into the streamwise direction. In particular, dw/dx appears to be an integral part of the tilting process. Low-speed streaks also appear to be important. We examine surface textures that could interfere with the spreading process including riblets and fins to interfere with the tilting process and dimples on the surface to lock in the spacing of low-speed streaks.

[1]Supported by AFOSR grant FA 9550-08-1-0453

3:02PM R20.00010 Similarities between statistically-stationary homogeneous shear turbulence and the logarithmic layer in channels[1] , SIWEI DONG, ATSUSHI SEKIMOTO, JAVIER JIMÉNEZ, Universidad Politécnica de Madrid — The rough independence of the logarithmic layer (LL) of wall-bounded turbulence from the details of the buffer and outer layers, suggests that the interaction of the turbulent fluctuations with the mean shear may be mimicked by statistically-stationary homogeneous shear turbulence(SS-HST) in a finite box. We study SS-HST in boxes for which the statistics best agree with those of the LL. Both flows share similar Corrsin shear parameters, and Reynolds-stress and vorticity anisotropies. Two-point correlation functions show that u and w are constrained by the simulation box and are respectively shorter and narrower for SS-HST than for the LL, but v and the vorticity are roughly of the same size in both flows when Re_λ is similar. The transient bursting of v in both flows is quite similar to the linear Orr mechanism, with time scales that are of the same order in both flows. In both cases, a streamwise velocity streak forms and breaks down quasi periodically, and the break down is accompanied by an enhanced flux of momentum, in the form of large-scale ejections and sweeps.

[1]Funded by the ERC Multiflow program and CSC

Tuesday, November 26, 2013 1:05PM - 3:15PM –

Session R21 Biofluids: Locomotion XII - Microswimmers and Bacteria III 316 - Jeffrey Guasto, Tufts University

1:05PM R21.00001 Ferromagnetic and antiferromagnetic order in bacterial vortex lattices , HUGO WIOLAND, FRANCIS G. WOODHOUSE, JÖRN DUNKEL, RAYMOND E. GOLDSTEIN, Department of Applied Mathematics and Theoretical Physics, Centre for Mathematical Sciences, University of Cambridge, GOLDSTEIN LAB TEAM — In conventional electronic materials, spins can organize into ordered phases that give rise to ferromagnetic or antiferromagnetic behavior. Here, we report similar observations in a completely different system: a suspension of swimming bacteria. When a dense *Bacillus subtilis* suspension is confined to a small circular chamber, it can spontaneously form a stable vortex ("spin") state that can persist for several minutes [Wioland et al., PRL **110**, 268102 (2013)]. By coupling up to 100 such chambers in microfluidic devices, we are able to realize bacterial spin lattices of different geometries. Depending on that geometry and the effective coupling strength between neighboring vortices, we observe the formation of stable "antiferromagnetic" and "ferromagnetic" bacterial vortex states, that appear to be controlled by the subtle competition between bacterial boundary layer flows and bulk dynamics.

1:18PM R21.00002 Magnetic Control of Rigid Achiral Microswimmers , U CHEANG, Drexel University, FARSHAD MESHKATI, HENRY FU, University of Nevada, MINJUN KIM, Drexel University — We report control of rigid achiral microswimmers in low Reynolds number environments. A rotating magnetic field was used to actuate the microswimmers wirelessly by rotating the microswimmers, which produces propulsion. Previous magnetically actuated microswimmers in bulk fluids have been designed with either flexibility or chiral geometry; we show that simpler geometries with neither flexibility nor chirality can produce propulsion. The microswimmer consists of three magnetic beads conjugated using avidin-biotin linkages into an arc formation. We designed a magnetic field generator consisting of electromagnetic coils arranged in an approximate Helmholtz configuration. A highspeed camera provided realtime imaging of the microswimmers' motion in a PDMS chamber. The rigidity of the microswimmer was characterized by tracking the position of the individual beads and calculating their relative distances. As a function of field strength and rotation frequency, we observed changes in the rotational axis of the microswimmers and the corresponding effects on their velocities. The achiral microswimmers exhibited active propulsion and were controllable in both speed and direction, which demonstrates the possibility for future biomedical applications such as drug delivery.

1:31PM R21.00003 Direct evidence of flagellar synchronization through hydrodynamic interactions , DOUGLAS BRUMLEY, MARCO POLIN, KIRSTY WAN, RAYMOND GOLDSTEIN, DAMTP, University of Cambridge — Eukaryotic cilia and flagella exhibit striking coordination, from the synchronous beating of two flagella in *Chlamydomonas* to the metachronal waves and large-scale flows displayed by carpets of cilia. However, the precise mechanisms responsible for flagellar synchronization remain unclear. We perform a series of experiments involving two individual flagella in a quiescent fluid. Cells are isolated from the colonial alga *Volvox carteri*, held in place at a fixed distance d, and oriented so that their flagellar beating planes coincide. In this fashion, we are able to explicitly assess the role of hydrodynamics in achieving synchronization. For closely separated cells, the flagella are capable of exhibiting a phase-locked state for thousands of beats at a time, despite significant differences in their intrinsic frequencies. For intermediate values of d, synchronous periods are interrupted by brief phase slips, while for $d \gg 1$ the flagellar phase difference drifts almost linearly with time. The coupling strength extracted through analysis of the synchronization statistics exhibits excellent agreement with hydrodynamic predictions. This study unambiguously reveals that flagella coupled only through hydrodynamics are capable of exhibiting robust synchrony.

1:44PM R21.00004 Feeding of swimming *Paramecium* with fore-aft asymmetry in viscous fluid , PENG ZHANG, SAIKAT JANA, Department of Engineering Science and Mechanics, Virginia Tech, MATTHEW GIARRA, PAVLOS VLACHOS, Department of Mechanical Engineering, Virginia Tech, SUNGHWAN JUNG, Department of Engineering Science and Mechanics, Virginia Tech — Swimming behaviours and feeding efficiencies of *Paramecium Multimicronucleatum* with fore-aft asymmetric body shapes are studied experimentally and numerically. Among various possible swimming ways, ciliates typically exhibit only one preferred swimming directions in favorable conditions. Ciliates, like *Paramecia*, with fore-aft asymmetric shapes preferably swim towards the slender anterior while feeding fluid to the oral groove located at the center of the body. Since both feeding and swimming efficiencies are influenced by fluid motions around the body, it is important to reveal the fluid mechanics around a moving object. Experimentally, μ-PIV methods are employed to characterize the source-dipole streamline patterns and fluid motions around *Paramecium*. Numerical simulations by boundary element methods are also used to evaluate surface stresses and velocities, which give insights into the efficiencies of swimming and feeding depending on body asymmetry. It is concluded that a slender anterior and fat posterior increases the combined efficiency of swimming and feeding, which matches well with actual shapes of *Paramecium*. Discrepancies between experiments and simulations are also discussed.

1:57PM R21.00005 Bacterial motility near crude oil and water interface[1] , JOMAYRA E. SÁNCHEZ RODRÍGUEZ, University of Puerto Rich Mayaguez Campus, MEHDI MOLAEI, JIAN SHENG, Texas Tech University — Study of biodegradation of crude oil by microbes requires profound understanding of their movement near oil-water interface as well as in/out of phase movement. Bacterial motilities are known to be modified by the presence of an interface through hydrodynamic interactions in addition to the chemotactic behavior towards the oil phase. Using digital holographic microscopy and phase contrast microscopy, we study locomotion of *Pseudomonas* sp (P62), a well-known hydrocarbon degrader under various chemo- and mechano-environmental conditions. Baseline experiments have been performed at different nutrient levels and Ion levels to identify effects of chemical environment on cell motility. Utilizing novel microfluidics and surface functionalization, we have established a stable vertical oil-water interface between top and bottom surfaces of the microfluidics, which allow clear optical access to observe bacterial movement near the interface. Three-dimensional trajectories of bacteria, obtained by analyzing recorded by digital holography microscopy, enable us to characterize bacterial swimming and orientation near interfaces. Chemotaxis velocity and swimming induced dispersion are measured directly as well as cell concentration distributions with respect to the distance to the interface.

[1]NIH, NSF, GoMRI

2:10PM R21.00006 The phylogeny of swimming kinematics: The environment controls flagellar waveforms in sperm motility , JEFFREY GUASTO, Tufts University, LISA BURTON, MIT, RICHARD ZIMMER, UCLA, ANETTE HOSOI, ROMAN STOCKER, MIT — In recent years, phylogenetic and molecular analyses have dominated the study of ecology and evolution. However, physical interactions between organisms and their environment, a fundamental determinant of organism ecology and evolution, are mediated by organism form and function, highlighting the need to understand the mechanics of basic survival strategies, including locomotion. Focusing on spermatozoa, we combined high-speed video microscopy and singular value decomposition analysis to quantitatively compare the flagellar waveforms of eight species, ranging from marine invertebrates to humans. We found striking similarities in sperm swimming kinematics between genetically dissimilar organisms, which could not be uncovered by phylogenetic analysis. The emergence of dominant waveform patterns across species are suggestive of biological optimization for flagellar locomotion and point toward environmental cues as drivers of this convergence. These results reinforce the power of quantitative kinematic analysis to understand the physical drivers of evolution and as an approach to uncover new solutions for engineering applications, such as micro-robotics.

2:23PM R21.00007 Flagellar Kinematics and Swimming Behavior of Algal Cells in Viscoelastic Fluids[1] , PAULO ARRATIA, University of Pennsylvania, JING YANG, JERRY GOLLUB, Haverford College — The motility behavior of microorganisms can be significantly affected by the rheology of their fluidic environment. In this talk, we experimentally investigate the effects of fluid elasticity on both the flagella kinematics and swimming dynamics of the microscopic alga *Chlamydomonas reinhardtii*. We find that the flagellar beating frequency and wave speed are both enhanced by fluid elasticity. Interestingly, the swimming speeds during the alga power and recovery strokes are enhanced by fluid elasticity for De$>$1. Despite such enhancements, however, the alga net forward speed is hindered by fluid elasticity by as much as 30% compared to Newtonian fluids of similar shear viscosities. The motility enhancements could be explained by the mechanism of stress accumulation in the viscoelastic fluid.

[1]This work was supported by the National Science Foundation - DMR-1104705.

2:36PM R21.00008 Fabrication and calibration of sensitively photoelastic biocompatible gelatin spheres , HENRY FU, ERICSON CENICEROS, ZEPHYR MCCORMICK, University of Nevada, Reno — Photoelastic gelatin can be used to measure forces generated by organisms in complex environments. We describe manufacturing, storage, and calibration techniques for sensitive photoelastic gelatin spheres to be used in aqueous environments. Calibration yields a correlation between photoelastic signal and applied force to be used in future studies. Images for calibration were collected with a digital camera attached to a linear polariscope. The images were then processed in Matlab to determine the photoelastic response of each sphere. The effect of composition, gelatin concentration, glycerol concentration, sphere size, and temperature were all examined for their effect on signal response. The minimum detectable force and the repeatability of our calibration technique were evaluated for the same sphere, different spheres from the same fabrication batch, and spheres from different batches. The minimum force detectable is 10 μN or less depending on sphere size. Factors which significantly contribute to errors in the calibration were explored in detail and minimized.

2:49PM R21.00009 Self-propelled Metallic Microrods by Ultrasonic Standing Waves , MAURICIO HOYOS, BNRS-ESPCI, SUZANNE AHMED, WEI WANG, THOMAS MALLOUK, Penn State University, DEPT OF CHEMISTRY TEAM, LAB PMMH UMR7636 CBRS TEAM — Particulate materials like rigid particles, cells, bacteria, vesicles, or metallic micro rods can be manipulated in a resonator by ultrasonic standing waves. In a resonator, an acoustic force makes species to migrate either toward the nodes or antinodes depending on the acoustic properties of particles. The acoustic force depends on particles volume, acoustic energy and on the acoustic contrast factor. The latter is a function of particle and fluid acoustic impedances. Acoustic impedance is defined as the product between the density and the sound velocity of a material. The acoustic manipulation has been accomplished mostly in microfluidic devices for separating blood cells from lipids, for driving air bubbles or for generating micro-aggregates of cells. The range of frequencies used is between 0.5 and 10MHz. In this presentation we shall focus on a new phenomenon we called self-acoustophoresis consisting on generating very high speed displacements of metallic microrods (gold, ruthenium) suspended in water; we shall s show how ultrasonic standing waves can be used for generating high speed rotation ofindividual as well as micro rod aggregates.this manipulation opens new possibilities to drug delivery using micro rods as conveyers.

3:02PM R21.00010 Active Motion Control of Tetrahymena pyriformis by Galvanotaxis and Geotaxis[1] , JIHOON KIM, DOYOUNG BYUN, Sungkyunkwan University, MIN JUN KIM, Drexel University — Recently, there has been increasing interest in the swimming behavior of microorganisms and biologically inspired micro-robots. These microorganisms naturally accompanied by complex motions. Therefore it is important to understand the flow characteristics as well as control mechanisms. One of eukaryotic cells, the protozoa are a diverse group of unicellular organisms, many of which are motile cilia. Motile cilia are cover on the surface of cell in large numbers and beat in oriented waves. Sequential beating motions of a single cilium form metachronal strokes, producing a propagation wave, and therefore the body is achieved propulsion force. So preliminary studies are achieved to understand the flow induced by swimming microorganisms. Based on hydrodynamic results, the follow study of a few micro-scale protozoa cell, such as the Tetrahymena pyriformis, has provided active or passive control into several external stimuli. In typical control methods, the galvanotaxis and geotaxis were adopted active and passive control, respectively. The validation of galvanotaxis is used DC and AC voltage. In terms of geotaxis, corrugated microstructures were used to control in the microchannel.

[1]This research was supported by the Ministry of Education, Science and Technology (MEST, 2011-0016461), National Science Foundation (NSF) CMMI Control Systems Program (#1000255) and Army Research Office (W911NF-11-1-0490).

Tuesday, November 26, 2013 1:05PM - 3:15PM –

Session R22 DFD Minisymposium: Frontiers in Combustion Physics II 317 - Javier Urzay, Stanford University

1:05PM R22.00001 Mixing in combustion[1] , PAUL DIMOTAKIS, California Institute of Technology — Mixing of reactants represents an important element in both non-premixed and premixed turbulent combustion. In non-premixed combustion, molecular mixing is a necessary first step that brings reactants together. In premixed combustion with local flame extinction and reignition, turbulent mixing of hot products with as-yet unburnt fluid is important to combustion behavior. The discussion on mixing will cover the role of entrainment, effects of Reynolds number and the mixing transition, effects of Schmidt number and gas- vs. liquid-phase reacting flows, heat release, Damkoehler-number (finite kinetic-rate) effects, and Mach-number effects.

[1]Support by the Air Force Office of Scientific Research

1:31PM R22.00002 Highly Turbulent Counterflow Flames: A Laboratory Scale Benchmark for Practical Combustion Systems , ALESSANDRO GOMEZ, Yale Center for Combustion Studies, Department of Mechanical Engineering and Materials Science, Yale University, New Haven, CT 06520-8286, USA — Since the pioneering work of Weinberg's group at Imperial College in the '60s, the counterflow system has been the workhorse of laminar flame studies. Recent developments have shown that it is also a promising benchmark for highly *turbulent* ($Re_t \sim 1000$) nonpremixed and premixed flames of direct relevance to gasturbine combustion. Case studies will demonstrate the versatility of the system in mimicking real flame effects, such as heat loss and flame stratification in premixed flames, and the compactness of the combustion region. The system may offer significant advantages from a computational viewpoint, including: a) aerodynamic flame stabilization near the interface between the two opposed jets, with ensuing simplifications in the prescription of boundary conditions; b) a fiftyfold reduction of the domain of interest as compared to conventional nonpremixed jet flames at the same Reynolds number; and c) millisecond mean residence times, which is particularly useful for DNS/LES computational modeling, and for soot suppression in the combustion of practical fuels.

In collaboration with Bruno Coriton and Jonathan Frank, Combustion Research Facility, Sandia National Laboratories, Livermore, CA 94551, USA.

1:57PM R22.00003 Supersonic combustion[1] , MIRKO GAMBA, University of Michigan — Combustion in the supersonic regime presents several challenges over what the low-speed counterpart admits. Here we will review some of these challenges, and we will describe some of the key features of one of the canonical flow fields in supersonic combustion: the reacting transverse jet in a supersonic crossflow (JISCF). From a practical standpoint, the key challenges that limit our control of this combustion regime are fast mixing, robust flame holding and stability. In turn, these aspects are controlled by the complex effects introduced by chemistry, compressibility, shocks and shock/flow interactions, turbulence and the underlying coupling among them. Some of their properties will be discussed here. In particular, for a JISCF in a Mach 2.4 high enthalpy crossflow, the reaction zone structure, its dependence on near-wall events, boundary layer, and shock/boundary layer interaction will be described. We will demonstrate the paramount importance of the coupling between boundary layers and compressibility to provide mechanisms for flame stabilization at the wall. Mixing characteristics, overall structure, and the link to global parameters (momentum flux, velocity and density ratios) that characterize the JISCF, and possibly free shear supersonic flows in general, will also be highlighted from non-reacting experiments.

[1]Sponsored by DoE PSAAP at Stanford University

2:23PM R22.00004 Kinetic Modeling of Low-Temperature Plasma Assisted Combustion , IGOR ADAMOVICH, Department of Mechanical and Aerospace Engineering, The Ohio State University, Columbus, OH 43210 — Quantitative insight into kinetics of low-temperature plasma assisted fuel oxidation and ignition would be impossible without kinetic modeling. The principal challenges in development of a predictive kinetic model of nonequilibrium plasmas sustained in fuel-air mixtures include (i) lack of "conventional" chemical kinetics mechanisms validated at low temperatures, (ii) lack of data on rates and products of reactions of excited species generated in the plasma, some of which are not well understood, and their coupling with fuel-air plasma chemistry, and (iii) scarcity of data obtained in well-characterized plasma-assisted combustion experiments, which can be used for model validation. "Conventional" combustion chemistry mechanisms have been developed for relatively high temperature conditions. Their applicability at temperatures below ignition temperature, common in plasma assisted combustion environments, needs to be assessed to determine if they can be used as a basis for a plasma-assisted combustion chemistry mechanism. This requires time-resolved measurements of radical species concentrations during low-temperature fuel oxidation, when an initial pool of primary radicals (O, H, and OH) is generated in the plasma, such as in the late afterglow of an electric discharge. This allows isolating relatively slow "conventional" low-temperature fuel oxidation reactions triggered by the radicals from the reactions of excited species generated in the discharge, which decay relatively rapidly. Kinetic modeling calculations demonstrated that some of the existing combustion mechanisms provide good agreement with the experimental data taken in lean H_2-, CH_4-, and C_2H_4-air mixtures at low temperatures, while data taken in C_3H_8-air are not reproduced by any of the mechanisms tested. A complementary approach is to focus on kinetics of "rapid" reactions of electronically and vibrationally excited species in the electric discharge, as well as oxygen dissociation by electron impact, and their effect on production of radicals in the early afterglow. These experiments provide key data on coupling of molecular energy transfer processes in the plasma with "conventional" chemical reactions. Time-resolved and spatially-resolved measurements of temperature, vibrational and electronic levels populations, and radical species concentrations are critical for characterization of the nonequilibrium reacting mixture at these conditions. Kinetic modeling of recent experiments in a diffuse filament, nanosecond pulse electric discharges in air suggest that the role of electronically excited N_2* molecules on chemical reactions in the afterglow, such as NO generation reactions, has been significantly underestimated in the past. Further experiments in fuel-air mixtures are expected to provide additional data on the role of these excited species on low-temperature fuel-air chemistry.

2:49PM R22.00005 Manipulating Flames with AC Electric Fields , KYLE BISHOP, Department of Chemical Engineering, Pennsylvania State University — Time-oscillating electric fields applied to plasmas present in flames create steady flows of gas capable of shaping, directing, enhancing, or even extinguishing flames. Interestingly, electric winds induced by AC electric fields can be stronger that those due to static fields of comparable magnitude. Furthermore, unlike static fields, the electric force due to AC fields is localized near the surface of the flame. Consequently, the AC response depends only on the local field at the surface of the flame - not on the position of the electrodes used to generate the field. These results suggest that oscillating electric fields can be used to manipulate and control combustion processes at a distance. To characterize and explain these effects, we investigate a simple experimental system comprising a laminar methane-air flame positioned between two parallel-plate electrodes. We quantify both the electric and hydrodynamic response of the flame as a function of frequency and magnitude of the applied field. A theoretical model shows how steady gas flows emerge from the time-averaged electrical force due to the field-induced motion of ions generated within the flame and by their disappearance by recombination. These results provide useful insights into the application of AC fields to direct combustion processes.

Tuesday, November 26, 2013 1:05PM - 3:02PM –

Session R23 Biofluids: Physiological VI - Experimental Studies in Blood Flows 318 - Alison Marsden, University of California, San Diego

1:05PM R23.00001 Dynamical systems characterization of inertial effects of fluid flow in a curved artery model under pulsatile flow forcing[1] , MICHAEL LEGGIERO, KARTIK V. BULUSU, MICHAEL W. PLESNIAK, The George Washington University — The main objective of this study was to examine inertial effects in a 180-degree model of curved arteries under pulsatile inflow conditions. Two-component, two-dimensional particle image velocimetery (2C-2D PIV) data were acquired upstream of and at several cross-sectional locations in the curved artery model. A blood-analog fluid comprised of 71% saturated sodium iodide solution, 28% glycerol and 1% distilled water (by volume) was subjected to multi-harmonic pulsatile inflow functions. First, signal time-lag was quantified by cross-correlating the input (voltage-time) supplied to a programmable pump and the output PIV (flow rate-time) measurements. The experiment was then treated as a linear, time-invariant system, and frequency response was estimated for phase shifts across a certain spectrum. Input-output signal dissimilarities were attributable to intrinsic inertial effects of flow. By coupling pressure-time and upstream flow rate-time measurements, the experiment was modeled using system identification methods. Results elucidate the role of inertial effects in fluid flow velocity measurements and the effect of these delays on secondary flow structure detection in a curved artery model.

[1]Supported by the NSF Grant No. CBET- 0828903 and GW Center for Biomimetics and Bioinspired Engineering.

1:18PM R23.00002 Pulse wave analysis in a 180-degree curved artery model: Implications under physiological and non-physiological inflows[1] , KARTIK V. BULUSU, MICHAEL W. PLESNIAK, The George Washington University — Systolic and diastolic blood pressures, pulse pressures, and left ventricular hypertrophy contribute to cardiovascular risks. Increase of arterial stiffness due to aging and hypertension is an important factor in cardiovascular, chronic kidney and end-stage-renal-diseases. Pulse wave analysis (PWA) based on arterial pressure wave characteristics, is well established in clinical practice for evaluation of arterial distensibility and hypertension. The objective of our exploratory study in a rigid 180-degree curved artery model was to evaluate arterial pressure waveforms. Bend upstream conditions were measured using a two-component, two-dimensional, particle image velocimeter (2C-2D PIV). An ultrasonic transit-time flow meter and a catheter with a MEMS-based solid state pressure sensor, capable of measuring up to 20 harmonics of the observed pressure waveform, monitored flow conditions downstream of the bend. Our novel continuous wavelet transform algorithm (PIVlet 1.2), in addition to detecting coherent secondary flow structures is used to evaluate arterial pulse wave characteristics subjected to physiological and non-physiological inflows. Results of this study will elucidate the utility of wavelet transforms in arterial function evaluation and pulse wave speed.

[1]Supported by NSF Grant No. CBET- 0828903 and GW Center for Biomimetics and Bioinspired Engineering.

1:31PM R23.00003 Secondary flow structures in the presence of Type-IV stent fractures through a bent tube model for curved arteries: Effect of circulation thresholding[1] , SHADMAN HUSSAIN, KARTIK V. BULUSU, MICHAEL W. PLESNIAK, The George Washington University — A common treatment for atherosclerosis is the opening of narrowed arteries resulting from obstructive lesions by angioplasty and stent implantation to restore unrestricted blood flow. "Type-IV" stent fractures involve complete transverse, linear fracture of stent struts, along with displacement of the stent fragments. Experimental data pertaining to secondary flows in the presence of stents that underwent "Type-IV" fractures in a bent artery model under physiological inflow conditions were obtained through a two-component, two-dimensional (2C-2D) PIV technique. Concomitant stent-induced flow perturbations result in secondary flow structures with complex, multi-scale morphologies and varying size-strength characteristics. Ultimately, these flow structures may have a role to play in restenosis and progression of atherosclerotic plaque. Vortex circulation thresholds were established with the goal of resolving and tracking iso-circulation secondary flow vortical structures and their morphological changes. This allowed for a parametric evaluation and quantitative representation of secondary flow structures undergoing deformation and spatial reorganization.

[1]Supported by NSF Grant No. CBET- 0828903 and GW Center for Biomimetics and Bioinspired Engineering.

1:44PM R23.00004 Setup of a Biomedical Facility to Study Physiologically Relevant Flow-Structure Interactions , FARAZ MEHDI, JIAN SHENG, Texas Tech University — The design and implementation of a closed loop biomedical facility to study arterial flows is presented. The facility has a test section of 25 inches, and is capable of generating both steady and pulsatile flows via a centrifugal and a dual piston pump respectively. The Reynolds and Womersley numbers occurring in major blood vessels can be matched. The working fluid is a solution of NaI that allows refractive index matching with both rigid glass and compliant polymer models to facilitate tomographic PIV and holographic PIV. The combination of these two techniques allows us to study both large scale flow features as well as flows very close to the wall. The polymer models can be made with different modulus of elasticity and can be pre-stressed using a 5-axis stage. Radially asymmetric patches can also be pre-fabricated and incorporated in the tube during the manufacturing process to simulate plaque formation in arteries. These tubes are doped with tracer particles allowing for the measurement of wall deformation. Preliminary flow data over rigid and compliant walls is presented. One of the aims of this study is to characterize the changes in flow as the compliancy of blood vessels change due to age or disease, and explore the fluid interactions with an evolving surface boundary.

1:57PM R23.00005 Effect of centrifugal forces on formation of secondary flow structures in a 180-degree curved artery model under pulsatile inflow conditions[1] , SHANNON CALLAHAN, ROSHAN SAJJAD, KARTIK V. BULUSU, MICHAEL W. PLESNIAK, The George Washington University — An experimental investigation of secondary flow structures within a 180-degree bent tube model of a curved artery was performed using phase-averaged, two-component, two-dimensional, particle image velocimetry (2C-2D PIV) under pulsatile inflow conditions. Pulsatile waveforms ranging from simple sinusoidal to physiological inflows were supplied. We developed a novel continuous wavelet transform algorithm (PIVlet 1.2) and applied it to vorticity fields for coherent secondary flow structure detection. Regime maps of secondary flow structures revealed new, deceleration-phase-dependent flow morphologies. The temporal instances where streamwise centrifugal forces dominated were associated with large-scale coherent structures, such as deformed Dean-, Lyne- and Wall-type (D-L-W) vortical structures. Magnitudes of streamwise and cross-stream centrifugal forces tend to balance during deceleration phases. Deceleration events were also associated with spatial reorganization and asymmetry in large-scale D-L-W secondary flow structures. Hence, the interaction between streamwise and cross-stream centrifugal forces that affects secondary flow morphologies is explained using a "residual force" parameter i.e., the difference in magnitudes of these forces.

[1]Supported by the NSF Grant No. CBET- 0828903 and GW Center for Biomimetics and Bioinspired Engineering.

2:10PM R23.00006 Distant downstream steady-state flow studies of a mechanical heart valve: PIV study of secondary flow in a model aortic arch[1] , BRANDON R. FIX, CHRISTOPHER J. POPMA, KARTIK V. BULUSU, MICHAEL W. PLESNIAK, The George Washington University — Each year, hundreds of thousands of aortic and mitral heart valves are replaced with prosthetic valves. In efforts to develop a valve that does not require lifelong anticoagulation therapy, previous experimental research has been devoted to analyzing the hemodynamics of various heart valve designs, limited to the flow up to only 2 diameters downstream of the valve. Two-component, two-dimensional (2C-2D) particle image velocimetry (PIV) was used in this study to examine secondary flow velocity fields in a curved tube modeling an aorta at five locations (0-, 45-, 90-, 135-, 180-degrees). A bileaflet valve, opened to 30-, 45-, and 59-degrees, and one (no-valve) baseline condition were examined under three steady flow inflows (Re = 218, 429, 634). In particular, variations in the two-dimensional turbulent shear stresses at each cross sectional plane were analyzed. The results suggest that bileaflet valves in the aortic model produce significant turbulence and vorticity up to 5.5 downstream diameters, i.e. up to the 90-degrees location. Expanding this research towards aortic heart valve hemodynamics highlights a need for additional studies extending beyond the typical few diameters downstream to fully characterize valvular function.

[1]Supported by the NSF Grant No. CBET- 0828903 and GW Center for Biomimetics and Bioinspired Engineering.

2:23PM R23.00007 Plasma protein induced clustering of red blood cells in micro capillaries[1] , CHRISTIAN WAGNER, MATHIAS BRUST, OTHMANE AOUANE, DANIEL FLORMANN, Experimental Physics, Saarland University, MARINE THIEBAUD, CLAUDE VERDIER, GWENNOU COUPIER, THOMAS PODGORSKI, CHAOUQI MISBAH, Laboratoire Interdisciplinaire de Physique, CNRS - UMR 5588, Université Grenoble I, HASSIB SELMI, Laboratoire d'Ingénierie Mathématique, Ecole Polytechnique de Tunisie — The plasma molecule fibrinogen induces aggregation of RBCs to clusters, the so called rouleaux. Higher shear rates in bulk flow can break them up which results in the pronounced shear thinning of blood. This led to the assumption that rouleaux formation does not take place in the microcapillaries of the vascular network where high shear rates are present. However, the question is of high medical relevance. Cardio vascular disorders are still the main cause of death in the western world and cardiac patients have often higher fibrinogen level. We performed AFM based single cell force spectroscopy to determine the work of separation. Measurements at low hematocrit in a microfluidic channel show that the number of size of clusters is determined by the adhesion strength and we found that cluster formation is strongly enhanced by fibrinogen at physiological concentrations, even at shear rate as high as 1000 1/s. Numerical simulations based on a boundary integral method confirm our findings and the clustering transition takes place both in the experiments and in the simulations at the same interaction energies. In vivo measurements with intravital fluorescence microscopy in a dorsal skin fold chamber in a mouse reveal that RBCs indeed form clusters in the micrcapillary flow.

[1]This work was supported by the German Science Foundation research imitative SFB1027

2:36PM R23.00008 Flow investigation in sidewall aneurysm model using a novel PIV multi-time-lag method , PIERRE BOUILLOT, OLIVIER BRINA, RAFIK OUARED, KARL-OLOF LOVBLAD, VITOR MENDES PEREIRA, Interventional Neuroradiology Unit, Service of Neuroradiology, University Hospitals of Geneva, MOHAMED FARHAT, Laboratory for Hydrolic Machines (LMH-IMHEF), Ecole Polytechnique Federale de Lausanne (EPFL), Avenue de Cour 33bis, CH-1007 Lausanne, Switzerland — The intracranial aneurysm (IA) lesion is one of the main causes of intracranial hemorrhage in productive population. It is well known that the hemodynamic factors have large impact on both the IAs rupture and treatment efficacy based on flow diverter stents. Precise experimental investigations of blood flow in IAs using particle imaging velocimetry (PIV) are therefore strongly required in order to validate clinical treatments based on computational and clinical flow assessment tools. Due to the large variations of flow velocities in IAs, a single PIV measurement with a unique time lag between two consecutive images cannot provide a good level of precision in all the measured volume. In this work, we implement an error analysis based on several PIV measurements with different time lags to ensure an optimal precision in the entire measurement volume. This PIV multi-time-lag method is applied on a sidewall IA model to investigate the effect of the inflow pulsatility. By comparing the flow patterns resulting from steady and unsteady inflows we point out important differences which could be involved in the IAs evolution. In particular, the blood transfer in the IA and the vortical structure are significantly modified when increasing the pulsatility compared to quasi-steady conditions.

2:49PM R23.00009 Effect of swirling inlet condition on the flow field in a stenosis phantom model , HOJIN HA, SANG JOON LEE, POSTECH, CENTER FOR BIOFLUID AND BIOMIMIC RESEARCH TEAM — The spiral blood flow effect in an axisymmetric stenosis model was experimentally investigated using particle image velocimetry velocity field measurement technique and streakline flow visualization. Spiral inserts with two different helical pitches (10D and 10/3D) were installed upstream of the stenosis to induce swirling flows. Results show that the spiral flow significantly reduces the length of recirculation flow and provokes early breakout of turbulent transition, but variation of swirling intensity does not induce significant changes of turbulence intensity. The present results about the spiral flow effects through the stenosis will contribute in achieving better understanding of the hemodynamic characteristics of atherosclerosis and in discovering better diagnosis procedures and clinical treatments.

Tuesday, November 26, 2013 1:05PM - 3:15PM –

Session R24 Biofluids: Physiological VII - Human Voice System Flows 319 - Lucy T. Zhang, Rensselaer Polytechnic Institute

1:05PM R24.00001 Study of dynamic fluid-structure coupling with application to human phonation[1] , SHAKTI SAURABH, JUSTIN FABER, DANIEL BODONY, University of Illinois at Urbana-Champaign — Two-dimensional direct numerical simulations of a compressible, viscous fluid interacting with a non-linear, viscoelastic solid are used to study the generation of the human voice. The vocal fold (VF) tissues are modeled using a finite-strain fractional derivative constitutive model implemented in a quadratic finite element code and coupled to a high-order compressible Navier-Stokes solver through a boundary-fitted fluid-solid interface. The viscoelastic solver is validated through in-house experiments using Agarose Gel, a human tissue simulant, undergoing static and harmonic deformation measured with load cell and optical diagnostics. The phonation simulations highlight the role tissue nonlinearity and viscosity play in the glottal jet dynamics and in the radiated sound.

[1]Supported by the National Science Foundation (CAREER award number 1150439).

1:18PM R24.00002 Fluid-Structure Interactions as Flow Propagates Tangentially Over a Flexible Plate with Application to Voiced Speech Production , ANDREA WESTERVELT, BYRON ERATH, Clarkson University — Voiced speech is produced by fluid-structure interactions that drive vocal fold motion. Viscous flow features influence the pressure in the gap between the vocal folds (i.e. glottis), thereby altering vocal fold dynamics and the sound that is produced. During the closing phases of the phonatory cycle, vortices form as a result of flow separation as air passes through the divergent glottis. It is hypothesized that the reduced pressure within a vortex core will alter the pressure distribution along the vocal fold surface, thereby aiding in vocal fold closure. The objective of this study is to determine the impact of intraglottal vortices on the fluid-structure interactions of voiced speech by investigating how the dynamics of a flexible plate are influenced by a vortex ring passing tangentially over it. A flexible plate, which models the medial vocal fold surface, is placed in a water-filled tank and positioned parallel to the exit of a vortex generator. The physical parameters of plate stiffness and vortex circulation are scaled with physiological values. As vortices propagate over the plate, particle image velocimetry measurements are captured to analyze the energy exchange between the fluid and flexible plate. The investigations are performed over a range of vortex formation numbers, and lateral displacements of the plate from the centerline of the vortex trajectory. Observations show plate oscillations with displacements directly correlated with the vortex core location.

1:31PM R24.00003 Flow in a Geometrically-Realistic, Vibrating Model of the Human Vocal Tract[1] , SCOTT THOMSON, JAYRIN SEEGMILLER, Brigham Young University — Airflow within the human vocal tract is an important component of voice quality. Understanding the nature of the airflow will help better understand voice production, potentially leading towards improved clinical diagnostics and treatments. An up-scaled experimental setup was developed to study three-dimensional flow features in a realistic model of the human larynx. The subglottal and supraglottal sections were made of clear silicone, with geometry derived from CT scan data. A cylindrically-shaped supraglottal section was also fabricated to compare flows with and without anatomically-accurate supraglottal sections. The glottal section consisted of two counter-rotating, mechanically-driven cams, covered by a silicone membrane, to approximate the alternating convergent-divergent profile of vibrating vocal folds. A mixture of water and glycerol was pumped through the system, the index of refraction matching that of the silicone for optical access into the sub- and supraglottal sections. Velocity fields throughout the glottal cycle were acquired using particle image velocimetry (PIV), giving particular attention to differences in flow features (e.g., jet skewing and axis switching) between models with CT-derived and cylindrically-shaped supraglottal geometry. In this presentation, the model design and characteristics will be given, and PIV flow results will be presented and discussed.

[1]Supported by NIH/NIDCD Grant R01 DC009616.

1:44PM R24.00004 Vocal Fold Pathologies and Three-Dimensional Flow Separation Phenomena[1] , ADAM G. APOSTOLI, KELLEY S. WEILAND, MICHAEL W. PLESNIAK, The George Washington University — Polyps and nodules are two different pathologies, which are geometric abnormalities that form on the medial surface of the vocal folds, and have been shown to significantly disrupt a person's ability to communicate. Although the mechanism by which the vocal folds self-oscillate and the three-dimensional nature of the glottal jet has been studied, the effect of irregularities caused by pathologies is not fully understood. Examining the formation and evolution of vortical structures created by a geometric protuberance is important, not only for understanding the aerodynamic forces exerted by these structures on the vocal folds, but also in the treatment of the above-mentioned pathological conditions. Using a wall-mounted prolate hemispheroid with a 2:1 aspect ratio in cross flow, the present investigation considers three-dimensional flow separation induced by a model vocal fold polyp. Building on previous work using skin friction line visualization, both the velocity flow field and wall pressure measurements around the model polyp are presented and compared.

[1]Supported by the National Science Foundation, Grant No. CBET-1236351 and GW Center for Biomimetics and Bioinspired Engineering (COBRE).

1:57PM R24.00005 Phonation aeroacoustic source strength estimation from sound pressure measurements , MICHAEL KRANE, ARL Penn State, ELIZABETH CAMPO, ARL Penn State (now at 3M), MICHAEL MCPHAIL, ARL Penn State — An experimental characterization of monopole and dipole source spectra in a model of the human upper airway is presented. The airway model is a life-scale, vertical, straight duct of square cross section, into which two model vocal folds are placed. Five microphones are positioned in the duct, two below and two above the vocal folds, with a fifth microphone placed at the "mouth." Time-mean subglottal pressure and volume flow rate are measured using a micromanometer and ball-element meter, respectively. In addition, pressure on either side of the model vocal folds are measured using Kulite XCS-093 pressure transducers, and the motion of the model vocal folds is captured using high-speed video. Cross-correlations between the microphone pairs are used to estimate the right- and left-running acoustic wave amplitude spectra above and below the model vocal folds. From these spectra and theoretical matching conditions at the inlet and outlet of the vocal fold constriction, source spectra are constructed. These are compared to independent estimates of source spectra obtained from the difference of the Kulite transducer pressures and the motion of the model vocal folds. Acknowledge support from NIH R01 DC005642 (MK, MM) and ARL E&F program (EC).

2:10PM R24.00006 Glottal aerodynamics in compliant, life-sized vocal fold models , MICHAEL MCPHAIL, GRANT DOWELL, MICHAEL KRANE, ARL Penn State — This talk presents high-speed PIV measurements in compliant, life-sized models of the vocal folds. A clearer understanding of the fluid-structure interaction of voiced speech, how it produces sound, and how it varies with pathology is required to improve clinical diagnosis and treatment of vocal disorders. Physical models of the vocal folds can answer questions regarding the fundamental physics of speech, as well as the ability of clinical measures to detect the presence and extent of disorder. Flow fields were recorded in the supraglottal region of the models to estimate terms in the equations of fluid motion, and their relative importance. Experiments were conducted over a range of driving pressures with flow rates, given by a ball flowmeter, and subglottal pressures, given by a micro-manometer, reported for each case. Imaging of vocal fold motion, vector fields showing glottal jet behavior, and terms estimated by control volume analysis will be presented. The use of these results for a comparison with clinical measures, and for the estimation of aeroacoustic source strengths will be discussed. Acknowledge support from NIH R01 DC005642.

2:23PM R24.00007 Control volume analyses of glottal flow using a fully-coupled numerical fluid-structure interaction model , JUBIAO YANG, Rensselaer Polytechnic Institute, MICHAEL KRANE, Pennsylvania State University, LUCY ZHANG, Rensselaer Polytechnic Institute — Vocal fold vibrations and the glottal jet are successfully simulated using the modified Immersed Finite Element method (mIFEM), a fully coupled dynamics approach to model fluid-structure interactions. A self-sustained and steady vocal fold vibration is captured given a constant pressure input at the glottal entrance. The flow rates at different axial locations in the glottis are calculated, showing small variations among them due to the vocal fold motion and deformation. To further facilitate the understanding of the phonation process, two control volume analyses, specifically with Bernoulli's equation and Newton's 2nd law, are carried out for the glottal flow based on the simulation results. A generalized Bernoulli's equation is derived to interpret the correlations between the velocity and pressure temporally and spatially along the center line which is a streamline using a half-space model with symmetry boundary condition. A specialized Newton's 2nd law equation is developed and divided into terms to help understand the driving mechanism of the glottal flow.

2:36PM R24.00008 Viscous Flow Structures Downstream of a Model Tracheoesophageal Prosthesis , FRANK HEMSING, BYRON ERATH, Clarkson University — In tracheoesophageal speech (TES), the glottis is replaced by the tissue of the pharyngeoesophageal segment (PES) as the vibrating element of speech production. During TES air is forced from the lungs into the esophagus via a prosthetic tube that connects the trachea with the esophagus. Air moving up the esophagus incites self-sustained oscillations of the surgically created PES, generating sound analogous to voiced speech. Despite the ubiquity with which TES is employed as a method for restoring speech to laryngectomees, the effect of viscous flow structures on voice production in TES is not well understood. Of particular interest is the flow exiting the prosthetic connection between the trachea and esophagus, because of its influence on the total pressure loss (i.e. effort required to produce speech), and the fluid-structure energy exchange that drives the PES. Understanding this flow behavior can inform prosthesis design to enhance beneficial flow structures and mitigate the need for adjustment of prosthesis placement. This study employs a physical model of the tracheoesophageal geometry to investigate the flow structures that arise in TES. The geometry of this region is modeled at three times physiological scale using water as the working fluid to obtain nondimensional numbers matching flow in TES. Modulation of the flow is achieved with a computer controlled gate valve at a scaled frequency of 0.22 Hz to mimic the oscillations of the PES. Particle image velocimetry is used to resolve flow characteristics at the tracheoesophageal prosthesis. Data are acquired for three cases of prosthesis insertion angle.

2:49PM R24.00009 Evaluation of Synthetic Self-Oscillating Models of the Vocal Folds[1] , ELIZABETH P. HUBLER, KELLEY S. WEILAND, ADRIENNE B. HANCOCK, MICHAEL W. PLESNIAK, The George Washington University — Approximately 30% of people will suffer from a voice disorder at some point in their lives. The probability doubles for those who rely heavily on their voice, such as teachers and singers. Synthetic vocal fold (VF) models are fabricated and evaluated experimentally in a vocal tract simulator to replicate physiological conditions. Pressure measurements are acquired along the vocal tract and high-speed images are captured at varying flow rates during VF oscillation to facilitate understanding of the characteristics of healthy and damaged VFs. The images are analyzed using a videokymography line-scan technique that has been used to examine VF motion and mucosal wave dynamics *in vivo*. Clinically relevant parameters calculated from the volume-velocity output of a circumferentially-vented mask (Rothenberg mask) are compared to patient data. This study integrates speech science with engineering and flow physics to overcome current limitations of synthetic VF models to properly replicate normal phonation in order to advance the understanding of resulting flow features, progression of pathological conditions, and medical techniques.

[1]Supported by the GW Institute for Biomedical Engineering (GWIBE) and GW Center for Biomimetics and Bioinspired Engineering (COBRE).

3:02PM R24.00010 Patient-Specific Computational Modeling of Human Phonation[1] , QIAN XUE, XUDONG ZHENG, University of Maine, UNIVERSITY OF MAINE TEAM — Phonation is a common biological process resulted from the complex nonlinear coupling between glottal aerodynamics and vocal fold vibrations. In the past, the simplified symmetric straight geometric models were commonly employed for experimental and computational studies. The shape of larynx lumen and vocal folds are highly three-dimensional indeed and the complex realistic geometry produces profound impacts on both glottal flow and vocal fold vibrations. To elucidate the effect of geometric complexity on voice production and improve the fundamental understanding of human phonation, a full flow-structure interaction simulation is carried out on a patient-specific larynx model. To the best of our knowledge, this is the first patient-specific flow-structure interaction study of human phonation. The simulation results are well compared to the established human data. The effects of realistic geometry on glottal flow and vocal fold dynamics are investigated. It is found that both glottal flow and vocal fold dynamics present a high level of difference from the previous simplified model. This study also paved the important step toward the development of computer model for voice disease diagnosis and surgical planning.

[1]The project described was supported by Grant Number ROlDC007125 from the National Institute on Deafness and Other Communication Disorders (NIDCD).

Tuesday, November 26, 2013 1:05PM - 2:23PM –

Session R25 Flow Control VIII: Surface Modulation, Interface Speed and Other Effects 320 - Beverley J. McKeon, California Institute of Technology

1:05PM R25.00001 The effect of mako sharkskin on laminar flow separation[1] , MICHAEL BRADSHAW, AMY LANG, The University of Alabama, PHILIP MOTTA, MARIA HABEGGER, The University of South Florida, ROBERT HUETER, Mote Marine Laboratory — Many animals possess effective 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 flexibility over the body. Previous biological findings have shown that the scales increase in flexibility from the leading to trailing edge over the pectoral fin as well as on various sections of the body. It is believed that the scale bristling may provide a mechanism for flow separation control that leads to decreased drag and increased maneuverability. This study involved testing a left pectoral fin of a shortfin mako shark as well as a cylinder with a sharkskin specimen applied circumferentially in a water tunnel facility under static, laminar conditions. Digital Particle Image Velocimetry (DPIV) was used to characterize the flow over the surfaces. Various Reynolds numbers were tested for both configurations, as well as several AOAs for the pectoral fin. The flow over the fin and cylinder were compared to a painted fin and a smooth PVC cylinder, respectively. The study found that the shark scales do, in fact, help to control flow separation. However, in order for the scales to bristle and trap the reversing flow, a certain magnitude of reversed flow and shear is required. This phenomenon seems to be most effective at near stall conditions and at higher Reynolds numbers.

[1]Support from REU grant 1062611 is greatfully acknowledged

1:18PM R25.00002 Controlling turbulent boundary layer separation using biologically inspired 2D transverse grooves[1] , AMY LANG, EMILY JONES, FARHANA AFROZ, University of Alabama — It is theorized that the presence of grooves, such as the sinusoidal ones found on dolphin skin or the cavities that form between bristled shark skin scales, can lead to induced boundary layer mixing and result in the control of turbulent boundary layer separation. To test this hypothesis, a series of water tunnel experiments using DPIV studied the characteristics of a flat plate turbulent boundary layer whereby a rotating cylinder was used to induce an adverse pressure gradient and resulting flow separation. The experiments were repeated with the use of a plate covered with two types of grooves, rectangular and sinusoidal, with a spacing of 2 mm in size. Flow similarity of the cavity flow was preserved between the experiments and flow over bristled shark skin scales. Both geometries resulted in a reduction of flow separation as measured by backflow coefficient. In addition, Reynolds stress profiles showed that as the pressure gradient was increased, the sinusoidal geometry outperformed the rectangular grooves in terms of increased mixing close to the wall. The sinusoidal plate also generated a lower momentum deficit within the boundary layer which would indicate a smaller drag penalty.

[1]Support from NSF grant CBET 0932352 and a UA Graduate Council Fellowship is gratefully acknowledged.

1:31PM R25.00003 A Novel Method to Induce Hydrodynamic Instability in Boundary Layer Flows[1] , MORTEZA GHARIB, DAVID JEON, FRANCISCO PEREIRA, BEVERLEY MCKEON, California Institute of Technology — We have developed a method to induce passive hydrodynamic displacement of boundary layer type flows by implementing spatially patterned hydrophobic patches in the form of bands and spots on the surface of a boundary layer plate. These patterns can be designed as parallel bands of a certain width, spacing and direction, or spots with a random or regular distribution of a certain shape, size and spatial density. We will present results from a series of experiments where the response of boundary layers in low to medium Reynolds number ranges to these spatial forcing will be demonstrated. Also, we will discuss potential use of this novel technique for drag reduction and separation delay applications where our technique could be used to replace riblets, trip wires and vortex generators.

[1]This work is supported by ONR- N00014-11-1-0031.

1:44PM R25.00004 Patterned Surface Roughness for Passive Transition Delay[1] , ROBERT DOWNS, JENS H.M. FRANSSON, Linné FLOW Centre, KTH Mechanics — Surface roughness is demonstrably detrimental to boundary-layer stability in many scenarios; it is now known that sensibly chosen roughness can also delay the onset of transition, resulting in a drag reduction. The latest part of an ongoing research effort[2] exploring the use of streamwise streaks to attenuate growth of forced disturbances, the present experiments employ a spatially periodic surface pattern to modify the flow in a flat plate boundary layer. With respect to conventional cylindrical surface roughness, the critical roughness-height-based Reynolds number of the surface pattern is improved. Tollmien–Schlichting waves are excited via suction and blowing at the wall, to form a well-controlled disturbance. A parametric study reveals that patterned roughness inhibits the growth of these T–S waves and increases the transition Reynolds number by 70% compared with the smooth plate reference case. Systematic changes to the pattern spacing demonstrate that the roughness can also accelerate the onset of transition.

[1]Support from the European Research Council (ERC) is acknowledged.
[2]Shahinfar et al. *Phys. Rev. Lett.* 109, 074501 (2012).

1:57PM R25.00005 Development of FDR-AF (Frictional Drag Reduction Anti-Fouling) Marine Coating[1] , INWON LEE, HYUN PARK, HO HWAN CHUN, GCRC-SOP, Pusan National University, GCRC-SOP TEAM — In this study, a novel skin-friction reducing marine paint has been developed by mixing fine powder of PEO(PolyEthyleneOxide) with SPC (Self-Polishing Copolymer) AF (Anti-Fouling) paint. The PEO is well known as one of drag reducing agent to exhibit Toms effect, the attenuation of turbulent flows by long chain polymer molecules in the near wall region. The frictional drag reduction has been implemented by injecting such polymer solutions to liquid flows. However, the injection holes have been a significant obstacle to marine application. The present PEO-containing marine paint is proposed as an alternative to realize Toms effect without any hole on the ship surface. The erosion mechanism of SPC paint resin and the subsequent dissolution of PEO enable the controlled release of PEO solution from the coating. Various tests such as towing tank drag measurement of flat plate and turbulence measurement in circulating water tunnel demonstrated over 10% frictional drag reduction compared with conventional AF paint.

[1]This work was supported by the National Research Foundation of Korea(NRF) grant funded by the Korea government(MSIP) through GCRC-SOP(No. 2011-0030013).

2:10PM R25.00006 ABSTRACT WITHDRAWN –

Tuesday, November 26, 2013 1:05PM - 2:49PM –

Session R26 Biofluids: Locomotion XIII - Bacteria/Flapping 321 - Maryam Jalali, Texas Tech University

1:05PM R26.00001 Turbulence-Copepod Interactions: Response of *Acartia tonsa* to Burgers Vortex , D.L. YOUNG, D.R. WEBSTER, J. YEN, Georgia Tech — Turbulence can affect the vertical position of copepods by altering their position, yet in situ studies suggest that, in many oceanic regimes, copepods alter their vertical position due to a behavioral response to turbulence. Numerous studies have examined copepod response to laminar flow fields, such as escaping from siphons and aggregating in thin layers. In contrast, little information exists on how they react to fine-scale turbulent fluid motions typically encountered in their natural marine environment. The hypothesis to be tested is that fine-scale turbulence alters copepod behavior, manifest as alterations in directed movement and changes in swimming kinematics. We present behavioral assays of the response of the coastal marine copepod, *Acartia tonsa*, to Burgers vortices. The rotation rate and axial strain rate of the Burgers vortices are specified to correspond to the vortices with the median dissipation rate in turbulent conditions typically encountered in coastal and near surface regions. The target conditions are defined by mean turbulent dissipation rates of 0.009 and 0.096 cm^2/s^3, respectively. The three dimensional flow field of each vortex treatment is quantified via tomographic-PIV, allowing for the analysis of copepod response to specific hydrodynamic cues such as deformation rate and vorticity. Copepod trajectories are analyzed in order to correlate the behavior responses (quantified as swimming kinematics) to the hydrodynamic sensory cue.

1:18PM R26.00002 Visualizing viral transport and host infection , KWANGMIN SON, MIT, JEFFREY GUASTO, Tufts university, ANDRES CUBILLOS-RUIZ, MIT, MATTHEW SULLIVAN, University of Arizona, ROMAN STOCKER, MIT, MIT TEAM — A virus is a non-motile infectious agent that can only replicate inside a living host. They consist of a <100 nm diameter capsid which houses their DNA, and a <20 nm diameter tail used to inject DNA to the host, which are classified into three different morphologies by the tail type: short tail ($\sim$ 10 nm, podovirus), rigid contractile tail ($\sim$ 100 nm, myovirus), or flexible noncontractile tail ($\sim$ 300 nm, siphovirus). Combining microfluidics with epifluorescent microscopy, we studied the simultaneous diffusive transport governing the initial encounter and ultimately the infection of a non-motile cyanobacteria host ($\sim$ 1 μm; prochlorococcus) and their viral (phage) counterparts in real time. This methodology allows us to quantify the virus-host encounter/adsorption dynamics and subsequently the effectiveness of various tail morphologies for viral infection. Viral transport and the role of viral morphology in host-virus interactions are critical to our understanding of both ecosystem dynamics and human health, as well as to the evolution of virus morphology.

1:31PM R26.00003 Bacterial locomotion, adsorption and growth over chemically patterned surfaces , MARYAM JALALI, MEHDI MOLAEI, JIAN SHENG, Texas Tech University — Complex dynamic interactions between bacteria and chemically patched interface that mimics the heterogeneous energy landscape of a real-life interfacial environment are studied in the paper. We explore effects of these spatially varying chemical patches on bacterial locomotion, adsorption, biofilm formation and the film growth rate. Using micro-fabrication and soft-lithography, we have fabricated PDMS microfluidic channels with a solid substrate covered by micro-scale chemical patches. Arrays of 2D geometries of characteristic scales varying from 10 to 50 μm are transferred onto a glass substrate by soft-lithography. The substrate is functionalized to generate alternating hydrophobic and hydrophilic regions and bonded with the channel. The 3D swimming characteristics near these surfaces, such as swimming velocity, linear and angular dispersions, are measured *in-situ* using 3D digital holographic microscopy. The observations are used to examine the mechanisms involved in adsorption and desorption of swimming bacteria onto the substrate. Long-term experiments are conducted to quantify the growth rate and structures of colony. A correlation between various length scales of the substrate and bacteria motility are observed.

1:44PM R26.00004 Shear alters motility of *Escherichia coli*[1] , MEHDI MOLAEI, MARYAM JALALI, JIAN SHENG, Texas Tech University — Understanding of locomotion of microorganisms in shear flows drew a wide range of interests in microbial related topics such as biological process including pathogenic infection and biophysical interactions like biofilm formation on engineering surfaces. We employed microfluidics and digital holography microscopy to study motility of *E. coli* in shear flows. We controlled the shear flow in three different shear rates: 0.28 s^{-1}, 2.8 s^{-1}, and 28 s^{-1} in a straight channel with the depth of 200 μm. Magnified holograms, recorded at 15 fps with a CCD camera over more than 20 minutes, are analyzed to obtain 3D swimming trajectories and subsequently used to extract shear responses of *E.coli*. Thousands of 3-D bacterial trajectories are tracked. The change of bacteria swimming characteristics including swimming velocity, reorientation, and dispersion coefficient are computed directly for individual trajectory and ensemble averaged over thousands of realizations. The results show that shear suppresses the bacterial dispersions in bulk but promote dispersions near the surface contrary to those in quiescent flow condition. Ongoing analyses are focusing to quantify effect of shear rates on tumbling frequency and reorientation of cell body, and its implication in locating the hydrodynamic mechanisms for shear enhanced angular scattering.

[1]NIH, NSF, GoMRI

1:57PM R26.00005 Rheological behaviour of a suspension of microswimmers varying in motor characteristics[1] , MAHESH TIRUMKUDULU, RICHA KARMAKAR, RANJIT GULVADY[2], K.V. VENKATESH, Indian Institute of Technology Bombay — A suspension of motile cells exhibits complex rheological properties due to their collective motion. We measure the shear viscosity of suspensions of Escherichia coli strains varying in motor characteristics such as duration of run and tumble. At low cell densities, all strains irrespective of their motor characteristics exhibiting a linear increase in viscosity with cell density suggesting that the cells behave as a suspension of rods with an effective aspect ratio set by the motor characteristics of the bacteria. As the cell density is increased beyond a critical value, the viscosity drops sharply signaling the presence of strongly coordinated motion among bacteria. The critical density depends not only on the magnitude of shear but also the motor characteristics of individual cells. High shear rate disrupts the coordinated motion reducing its behavior, once again, to a suspension of inactive particles.

[1]The authors acknowldege financial support from Department of Science and Technology, India
[2]Ranjit Gulvady is current pursuing his doctoral studies at NUS, Singapore

2:10PM R26.00006 Near wake features of a flying European Starling , ADAM KIRCHHEFER, GREGORY KOPP, University Of Western Ontario, ROI GURKA, BenGurion University — A great deal of research focusing on flapping wings has been motivated by their high performance capabilities, especially in low Reynolds number configurations where static wing performance typically suffers. The approaches to studying flapping wings have taken different forms. One form has been the systematic investigation of the parameters that influence the relationship between flapping wings and their wake. The other form, and the approach used in the present work, is the investigation of flapping wings in nature. While the earliest work on the flapping wings of animals consists of observations of bird flight by Leonardo DaVinci, advances in technology have allowed for quantitative measurements of the wake. The near wake of a freely flying European starling has been measured using high speed, time-resolved, particle image velocimetry, simultaneously with high speed cameras which imaged the bird. These have been used to measure the near wake two-dimensional velocity field that can be associated with the bird's location and wing configuration in an avian wind tunnel. Time series of the velocities have been expressed as composite wake plots, which depict segments of the wing beat cycle for various spanwise locations in the wake. Measurements indicate that downwash is not produced during the upstroke, suggesting that the upstroke does not generate lift. As well, the wake velocities imply the presence of streamwise vortical structures, in addition to tip vortices. These two characteristics indicate similarities between the wake of a bird and the wake of a bat.

2:23PM R26.00007 Unsteady Propulsors in Ground Effect[1] , KEITH MOORED, Lehigh University, DANIEL QUINN, PETER DEWEY, Princeton University, ALEXANDER SMITS, Princeton University, Monash University — Experimental and computational results are presented on an airfoil undergoing pitch oscillations in ground effect, that is, close to a solid boundary. The time-averaged thrust is found to increase monotonically as the mean position of the airfoil approaches the boundary while the propulsive efficiency stays relatively constant, showing that ground effect can enhance thrust at little extra cost for a pitching airfoil. Vortices shed into the wake form pairs rather than vortex streets, so that in the mean a momentum jet is formed that angles away from the boundary.

[1]Supported by the Office of Naval Research under Program Director Dr. Bob Brizzolara, MURI grant number N00014-08-1-0642.

2:36PM R26.00008 Robotic penguin-like propulsor with novel spherical joint , BASSEM SUDKI, MICHEL LAURIA, FLAVIO NOCA, University of Applied Sciences - hepia - Geneva, Switzerland — We have designed and manufactured an innovative spherical joint mechanism with three actuated degrees of freedom, aimed at mimicking a penguin shoulder and enabling a potential propulsion technology with high efficiency and maneuverability. In addition, the mechanism might also lead to propellers with directional thrusting capability. A parallel architecture was chosen for this type of mechanism in order to ensure rigidity as well high actuation frequencies and amplitudes. Indeed, as the motors are fixed, inertial forces are lower than for a serial robot. The resulting spherical parallel mechanism (SPM) with coaxial shafts was designed and manufactured with the following specifications: fixed center of rotation (spherical joint); working frequency of 2.5 Hz under charge; unlimited rotation about main axis; and arbitrary motion within a cone of 60 degrees. The equations for the inverse kinematics of the mechanism have been established and can yield the trajectories of each actuator for any desired motion applied to the oar or blade. The technology will be illustrated with preliminary experiments in a hydrodynamic channel at the University of Applied Sciences - hepia - Switzerland.

Tuesday, November 26, 2013 1:05PM - 3:15PM –

Session R28 Industrial Applications II Spirit of Pittsburgh Ballroom B/C - John R. Buchanan Jr., Bechtel Marine Propulsion Corporation, Bettis Atomic Power Laboratory

1:05PM R28.00001 Influence of Spatial Variations on the Flow Field and Power Production of a Model Wind Farm[1] , ANGELISSE RAMOS, NICHOLAS HAMILTON, DOMINIC DELUCIA, RAÚL BAYOÁN CAL, Portland State University — Wind tunnel experiments of a 4 $\times$ 3 model wind turbine array are carried out to understand impact on the flow field and turbulence statistics due to the changes in turbine spacing. Stereo particle Image Velocimetry (SPIV) is used to obtain measurements in dual planes, fore and aft of wind turbine models in the centerline of the array. Variations in turbulence statistics are assessed by altering the streamwise and spanwise spacing. Spacing schemes tested include permutations of streamwise spacing, $S_x = [3D, 6D]$, and spanwise spacing, $S_z = [1.5D, 3D]$, where D is the rotor diameter. Quantities in the mean kinetic energy equation are analyzed under these variations including the Reynolds stress tensor, $\langle u_i u_j \rangle$, kinetic energy flux, $-\langle uv \rangle U$, and turbulence production, $-\langle uv \rangle \partial U/\partial y$. Furthermore, the mechanical power is measured for these turbines reflecting the influence of spatial variations. The analysis has consequences on land use $versus$ power output.

[1]National Science Foundation: ECCS-1032647

1:18PM R28.00002 Wind Turbine Gust Prediction Using Remote Sensing Data , PAUL TOWERS, BRYN JONES, Department of Automatic Control and Systems Engineering, University of Sheffield — Offshore wind energy is a growing energy source as governments around the world look for environmentally friendly solutions to potential future energy shortages. In order to capture more energy from the wind, larger turbines are being designed, leading to the structures becoming increasingly vulnerable to damage caused by violent gusts of wind. Advance knowledge of such gusts will enable turbine control systems to take preventative action, reducing turbine maintenance costs. We present a system which can accurately forecast the velocity profile of an oncoming wind, given only limited spatial measurements from light detection and ranging (LiDAR) units, which are currently operational in industry. Our method combines nonlinear state estimation techniques with low-order models of atmospheric boundary-layer flows to generate flow-field estimates. We discuss the accuracy of our velocity profile predictions by direct comparison to data derived from large eddy simulations of the atmospheric boundary layer.

1:31PM R28.00003 Detecting Unsteady Blade Row Interaction in a Francis Turbine using a Phase-Lag Boundary Condition[1] , ALEX WOUDEN, JOHN CIMBALA, BRYAN LEWIS, Pennsylvania State University — For CFD simulations in turbomachinery, methods are typically used to reduce the computational cost. For example, the standard periodic assumption reduces the underlying mesh to a single blade passage in axisymmetric applications. If the simulation includes only a single array of blades with an uniform inlet condition, this assumption is adequate. However, to compute the interaction between successive blade rows of differing periodicity in an unsteady simulation, the periodic assumption breaks down and may produce inaccurate results. As a viable alternative the phase-lag boundary condition assumes that the periodicity includes a temporal component which, if considered, allows for a single passage to be modeled per blade row irrespective of differing periodicity. Prominently used in compressible CFD codes for the analysis of gas turbines/compressors, the phase-lag boundary condition is adapted to analyze the interaction between the guide vanes and rotor blades in an incompressible simulation of the 1989 GAMM Workshop Francis turbine using OpenFOAM. The implementation is based on the "direct-storage" method proposed in 1977 by Erdos and Alzner. The phase-lag simulation is compared with available data from the GAMM workshop as well as a full-wheel simulation.

[1]Funding provided by DOE Award number: DE-EE0002667.

1:44PM R28.00004 Modified Design of Hydroturbine Wicket Gates to Include Liquid Control Jets[1] , BRYAN LEWIS, JOHN CIMBALA, ALEX WOUDEN, Penn State University — With the ever-increasing penetration of alternative electricity generation, it is becoming more common to operate hydroturbines under off-design conditions in order to maintain stability in the electric power grid. Improving the off-design performance of these turbines is therefore of significant importance. As the runner blades of a Francis hydroturbine pass though the wakes created by the upstream guide vanes (wicket gates and stay vanes), they experience significant changes in the instantaneous values of absolute velocity, flow angle, and pressure. The concept of adding water jets to the trailing edge of the guide vanes is proposed as a method for reducing the dynamic load on the hydroturbine runner blades, as well as modifying the flow angle of the water entering the runner to improve turbine efficiency during off-design operation. In order to add water jets that are capable of turning the flow, a modified beveled trailing edge design is presented. Computational experiments show that a $\pm 5^\circ$ change in swirl angle is achievable with the new design, as well as up to 4% improvement in turbine efficiency during off-design operation. This correlates to an overall improvement in machine efficiency of up to 2%, when the losses through the jet channels are taken into account.

[1]Funding for this work was provided by the DOD, through the National Defense Science and Engineering Graduate (NDSEG) Fellowship, and the DOE, through the Penn State Hydropower Research Grant.

1:57PM R28.00005 Mixing and transport in a liquid metal electrode , DOUGLAS KELLEY, University of Rochester, DONALD SADOWAY, Massachusetts Institute of Technology — Adding large-scale energy storage to Earth's electrical grids would accommodate demand variations, reduce the need for gas-fired peakers, and enable broad deployment of wind and solar generation. Liquid metal batteries are currently being commercialized as a promising and economically viable technology for grid-scale storage. But because these batteries are entirely liquid, fluid flow and instabilities affect battery robustness and performance. We present ultrasound measurements of flow in a realistic liquid metal electrode. We find a critical electrical current density above which the convective flow organizes and gains speed, improving battery efficiency. We are also developing numerical models to simulate flow and transport in liquid metal batteries.

2:10PM R28.00006 Fluid flow in discrete fractures in Enhanced/Engineered Systems, consequences of interconnected fractures, buoyancy, and fracture roughness[1] , DON FOX, DONALD KOCH, JEFFERSON TESTER, School of Chemical and Biomolecular Engineering — In Enhanced/Engineered Geothermal Systems (EGS), fluid flow in the discrete fracture network governs how thermal energy is "farmed" from the systems. The flow is created by an injection and production well but apart from pressure driven flow, temperature gradients will also cause natural convection in the system. Due to the roughness and shearing of the fractures, the aperture of fracture varies spatially and has been shown to be self-affine. Fracture roughness can lead to flow channeling where most of the flow is conducted through a single pathway. Knowing the flow regime is also important in understanding the behavior of tracers that are injected into the system. The tracer's residence time distribution is used to determine characteristics of the fractures and how they are possibly connected. This presentation will focus on how one would model the fluid flow in EGS and the consequences caused by interconnection, fracture roughness, and buoyancy have on fluid flow, how energy is extracted, and the behavior of tracers.

[1]NSF Earth Energy IGERT

2:23PM R28.00007 POD-Based Model Reduction toward Efficient Simulation of Flow in NuclearReactor Components , MOHAMMAD AHMADPOOR, GREG BANYAY, SAGNIK MAZUMDAR, University of Pittsburgh, ANIRBAN JANA, Pittsburgh Super Computing Center, MARK KIMBER, JOHN BRIGHAM, University of Pittsburgh — The long-term objective of this research is reduced-order modeling (ROM) to simulate and understand the turbulent mixing inside the lower plenum of a Very High Temperature Reactor, while the present study focuses on confined isothermal jet flow. In general, two steps are required to generate a basis for a ROM: (1) acquisition of an ensemble of possible solution fields for the system; and (2) extracting key features of the ensemble to create the basis. Proper Orthogonal Decomposition (POD) is one approach for extracting features from an ensemble. For this work POD is used to capture the parametric variation of a flow with Reynolds (Re) number and time. Two approaches are considered for model reduction: (1) a regression-based approach, which does not keep the mathematical structure of the modeling, but rather uses interpolation and/or extrapolation to predict flow fields at different Re number or different times and (2) a Galerkin-projection approach in which the Navier-Stokes equations are projected onto the POD modes to obtain low-dimensional ordinary differential equations to represent the fluid flow under conditions outside of the original ensemble.

2:36PM R28.00008 Experimental study on spray characteristics of alternate jet fuels using Phase Doppler Anemometry[1] , KUMARAN KANNAIYAN, REZA SADR, Texas A&M University at Qatar — Gas-to-Liquid (GTL) fuels have gained global attention due to their cleaner combustion characteristics. The chemical and physical properties of GTL jet fuels are different from conventional jet fuels owing to the difference in their production methodology. It is important to study the spray characteristics of GTL jet fuels as the change of physical properties can affect atomization, mixing, evaporation and combustion process, ultimately affecting emission process. In this work, spray characteristics of two GTL synthetic jet fuels are studied using a pressure-swirl nozzle at different injection pressures and atmospheric ambient condition. Phase Doppler Anemometry (PDA) measurements of droplet size and velocity are compared with those of regular Jet A-1 fuel at several axial and radial locations downstream of the nozzle exit. Experimental results show that although the GTL fuels have different physical properties such as viscosity, density, and surface tension, among each other the resultant change in the spray characteristics is insignificant. Furthermore, the presented results show that GTL fuel spray characteristics exhibit close similarity to those of Jet A-1 fuel.

[1]Funded by Qatar Science and Technology Park

2:49PM R28.00009 Fouling of Air Cooled Condensers On the Air Side , HAZEL MARIE, Youngstown State University, NICHOLAS MATUNE, Babcock & Wilcox Company — As the electrical power demand increases and water resources become more limited, fouling on the air side of Air Cooled Condensers (ACC) is a growing concern. The objective of this study was to experimentally and computationally calculate the convection heat transfer coefficient for both a clean and fouled condenser. Bee pollen was selected as the experimental fouling particle, and engineering data for similar particles were used for the computational model. Both the experimental and computational results showing the negative impact fouling has a on the heat transfer will be presented.

3:02PM R28.00010 Reducing numerical costs for core wide nuclear reactor CFD simulations by the Coarse-Grid-CFD , MATHIAS VIELLIEBER, ANDREAS G. CLASS, Karlsruhe Institute of Technology — Traditionally complete nuclear reactor core simulations are performed with subchannel analysis codes, that rely on experimental and empirical input. The Coarse-Grid-CFD (CGCFD) intends to replace the experimental or empirical input with CFD data. The reactor core consists of repetitive flow patterns, allowing the general approach of creating a parametrized model for one segment and composing many of those to obtain the entire reactor simulation. The method is based on a detailed and well-resolved CFD simulation of one representative segment. From this simulation we extract so-called parametrized volumetric forces which close, an otherwise strongly under resolved, coarsely-meshed model of a complete reactor setup. While the formulation so far accounts for forces created internally in the fluid others e.g. obstruction and flow deviation through spacers and wire wraps, still need to be accounted for if the geometric details are not represented in the coarse mesh. These are modelled with an Anisotropic Porosity Formulation (APF). This work focuses on the application of the CGCFD to a complete reactor core setup and the accomplishment of the parametrization of the volumetric forces.

Tuesday, November 26, 2013 1:05PM - 3:15PM –

Session R30 Microfluids: Microchannels 408 - Sangjin Ryu, University of Nebraska, Lincoln

1:05PM R30.00001 Noninvasive Measurement of the Pressure Distribution in a Deformable Micro-Channel[1] , OZGUR OZSUN, VICTOR YAKHOT, KAMIL L. EKINCI, Boston University — Direct and noninvasive measurement of the pressure drop in test sections of a rigid micro-channel is a challenging task. In a micro-channel with compliant walls, however, it is possible to determine the pressure field under flow from the local deflection of the channel walls. Here, we present a robust analytical approach for determining the pressure distribution in a deformable micro-channel under flow. In this method, we first measure the channel deflection profile as a function of applied hydrostatic pressure; this initial measurement provides the constitutive curves for the deformable channel. We then match the deflection profile under flow to the constitutive curves, obtaining the hydrodynamic pressure distribution. We have tested and validated the developed mapping on planar micro-fluidic channels. This method remains accurate in a broad parameter space, and can find possible applications in microfluidics and for characterizing biological flows.

[1]We acknowledge generous support from the US NSF through Grant No. CMMI-0970071

1:18PM R30.00002 Flow rate–pressure drop relation for deformable shallow microfluidic channels[1] , IVAN C. CHRISTOV, Theoretical Division & Center for Nonlinear Studies, Los Alamos National Laboratory, VINCENT COGNET, École Normale Supérieure de Cachan, HOWARD A. STONE, Mechanical & Aerospace Engineering, Princeton University — Laminar flow in devices fabricated from PDMS causes deformation of the passage geometry, which affects the flow rate–pressure drop relation. Having an accurate flow rate–pressure drop relation for deformable microchannels is of importance given that the flow rate for a given pressure drop can be as much as 500% of the flow rate predicted by Poiseuille's law for a rigid channel. Gervais et al. [*Lab Chip* **6** (2006) 500] proposed a successful model of the latter phenomenon by heuristically coupling linear elasticity with the lubrication approximation for Stokes flow. However, their model contains a fitting parameter that must be found for each channel shape by performing an experiment. We present a perturbative derivation of the flow rate–pressure drop relation in a shallow deformable microchannel using Kirchoff–Love theory of isotropic quasi-static plate bending and Stokes' equations under a "double lubrication" approximation (i.e., the ratio of the channel's height to its width and of the channel's width to its length are both assumed small). Our result contains no free parameters and confirms Gervais et al.'s observation that the flow rate is a quartic polynomial of the pressure drop.

[1]ICC was supported by NSF Grant DMS-1104047 and the U.S. DOE through the LANL/LDRD Program; HAS was supported by NSF Grant CBET-1132835.

1:31PM R30.00003 Shear and Pressure Driven Flow in Microchannels , YOGESH JALURIA, Rutgers University — In many important circumstances, microchannel flows driven by moving surfaces that impart shear to the fluid and by an imposed pressure difference across the channel are of interest. The pressure may aid or oppose the flow due to the moving surface. One such problem is the optical fiber coating process, where the entrance of the moving fiber into a reservoir of fluid, as well as its exit, results in shear driven flow in microchannels. An additional aiding or opposing pressure head is also usually applied. The transport processes influence the resulting coating very substantially. This paper discusses the basic considerations that arise in such processes, particularly the resulting flow and the menisci that are observed at the inlet and outlet regions of the two microchannels. Visualization has been an important approach to the basic understanding of these flows. Detailed flow and thermal transport results are often obtained by numerical modeling. Another important circumstance is the pressure rise in the channel for narrowing flow domains, such as those employed in dies and extruders. It is found that, in practical problems, high pressures are generated that oppose the shear effects. Then the resulting transport is affected by both shear and pressure. On the other hand, cooling of electronic systems often employs pressure-driven microchannel flows. Comparisons between the results obtained for these different flow situations indicate many interesting features, which are discussed in terms of the basic mechanisms.

1:44PM R30.00004 Momentum and mass transport over a superhydrophobic bubble mattress: the influence of interface geometry , PEICHUN AMY TSAI, A. SANDER HAASE, ELIF KARATAY, ROB LAMMERTINK, University of Twente, SOFT MATTER, FLUIDICS AND INTERFACES GROUP TEAM — We numerically investigate the influence of interface geometry on momentum and mass transport on a partially slippery bubble mattress. The bubble mattress, forming a superhydrophobic substrate, consists of an array of slippery (shear-free) gas bubbles with (no-slip) solids walls in between. We consider steady pressure-driven laminar flow over the bubble mattress, with a solute being supplied from the gas bubbles. The results show that solute transport can be enhanced significantly due to effective slippage, compared to a fully saturated no-slip wall. The enhancement depends on the interface geometry of the bubble mattress, i.e. on the bubble size, protrusion angle, and surface porosity. In addition, we demonstrate that the mass transfer enhancement disappears below a critical bubble size. The effective slip vanishes for very small bubbles, whereby interfacial transport becomes diffusion dominated. For large bubbles, solute transport near the interface is greatly enhanced by convection. The results provide insight into the optimal design of ultra-hydrophobic bubble mattresses to enhance both momentum and mass transport.

1:57PM R30.00005 ABSTRACT WITHDRAWN –

2:10PM R30.00006 Spontaneous oscillations in simple fluid networks , DEBORAH HELLEN, ERIKA WEILER, Olin College, NATHAN KARST, Babson College, JOHN GEDDES, BRIAN STOREY, Olin College — Nonlinear phenomena including multiple equilibrium states and spontaneous oscillations can occur in fluid networks containing multiple fluid phases. Such behavior might be attributed to the complicated geometry of the network, the complex rheology of the constituent fluids, or, in the case of microvascular blood flow, biological control. However, the simplest networks containing two miscible Newtonian fluids of differing viscosities are found to exhibit these non-linear phenomena. We use a combination of analytic and numerical techniques to identify and track saddle-node and Hopf bifurcations through the large parameter space. The model predictions show regions of sustained spontaneous oscillations and we investigate the sensitivity of these oscillations to changes in the viscosity contrast and network geometry. The model predictions are used to guide ongoing experimental which has confirmed the existence of such oscillations.

2:23PM R30.00007 Developing and testing models for flow in microdevices , LAURA DICKINSON, University of Bristol, JONATHAN KOBINE, University of Dundee — Microtechnology has developed faster than the corresponding theory describing the physics behind it. The continuum assumptions of classical fluid mechanics break down at the smallest lengthscales; however, Stokes and Poiseuille flow applies for liquid flow in microchannels, and there is merit in using the Navier-Stokes equations as a starting point from which accurate yet parsimonious models of flow in microdevices can be developed. We derive such models for the pressure drop across and the flux through a generic 2D microvalve, which show how each varies with valve opening (the characteristic curve). Eg, the nondimensional pressure drop P in terms of scaled valve opening H can be expressed as

$$P = 1 - 2\frac{(1-H^6)^{\frac{1}{2}} - 1}{H^6}.$$

We verify these models using a finite-element Navier-Stokes solver. Valves with a "long seat" and laminar flows are modeled accurately. We make additions to our models to capture the effects of flow at higher Re. The results of our simulations highlight flow behavior which is interesting to practitioners in the field of applied microfluidics; sharp valve edges produce separation and recirculation, small valve openings produce two jets which recombine in the valve exit. These phenomena give rise to the (turbulent) mixing which is desired but not easily achieved in microdiagnosics.

2:36PM R30.00008 On the Effects of 3D Field Focusing at a Heterogeneous Permselective Surface on Concentration Polarization , YOAV GREEN, GILAD YOSSIFON, Technion-Israel Institute of Technology — Understanding the effects of 2D and 3D geometric field focusing effects at the interface of a microreservoir-nanochannel system is of much importance in the growing field of electrokinetics and microfluidics. Such effects have been used in numerous and varying experimental systems but little theoretical work has been conducted to better understand these effects quantitatively. Previous studies made a number of oversimplifying assumptions regarding the geometry of the microreservior and its effects on concentration polarization so that the solution was valid only at certain limits. A 3D analytical solution is derived for the concentration and electric potential for an electrolyte undergoing concentration polarization. The effects of the both the microreservoir's and the permselective interface's geometry are investigated. It is shown that limiting current transported through the permselective surface is not only a function of the area but is strongly geometry dependent (i.e. rectangular or square surface). Additionally, it is shown that there is an amplification of the current density with increased field focusing effects which stands in agreement with previous experimental results.

2:49PM R30.00009 The role of erythrocyte size and shape in microchannel fluid dynamics[1] , KATHRYN FINK, JACOBO PAREDES, DORIAN LIEPMANN, University of California, Berkeley — The unique properties of blood flow in microchannels have been studied for nearly a century; much of the observed blood-specific dynamics is attributed to the biconcave shape of red blood cells. However, for almost twice as long biologists have observed and characterized the differences in the size and shape of red blood cells among vertebrates. With a few exceptions, mammals share the denucleated biconcave shape of erythrocytes but vary in size; oviparous vertebrates have nucleated ovoid red blood cells with size variations of a full order of magnitude. We utilize micro-PIV to analyze blood flow of vertebrate species in microchannels, with a focus on understanding how erythrocyte size and shape alter the cell-free layer and velocity profile of whole blood. The results offer insight into the Fahraeus-Lindqvist effect and the selection of animal blood for the design and evaluation of biological microfluidic devices.

[1]This material is based upon work supported by the National Science Foundation Graduate Research Fellowship under Grant No. DGE 1106400

3:02PM R30.00010 Deformations of micro-capsules through channels with corners[1] , LUCA BRANDT, LAILAI ZHU, KTH Mechanics, Linne flow centre — Deformable micro-particles moving in confined geometries are ubiquitous in nature from biological cells to biomedical and industrial applications such as synthetic capsules. Previous studies have demonstrated rich and complex behaviors of capsules and vesicles in the 2D Poiseuille flow, in a duct and in a pipe. Nevertheless, micro-particles commonly need to go through asymmetric geometries, for example a corner. Here we numerically study the dynamics of a Neo-Hookean capsule transported in a 3D channel with a 90 degree straight corner. We use the boundary integral method to solve the Stokes flow, accelerated by the general geometry Ewald method (GGEM) implemented in the framework of the general Navier-Stokes solver NEK5000 based on the spectral element method. A global spectral description utilizing spherical harmonics is incorporated to resolve simultaneously the membrane dynamics. We analyze the trajectory and deformation of the capsule, as well as the variation of area, velocity, principle stress and elastic energy. The influence of the capsule elasticity and wall confinement is also investigated. Finally, the flow in a smooth corner is simulated and compared with the straight counterpart, to provide hints for the design of micro-devices.

[1]We acknowledge the funding from VR (Swedish research council) and computer time from SNIC

Tuesday, November 26, 2013 1:05PM - 3:41PM –

Session R31 Focus Session: Structure of Turbulent/Non-Turbulent Interface 402 - Carlos da Silva, IST-Technical University of Lisbon

1:05PM R31.00001 Multiscale geometry, scaling and fluxes at the turbulent/non-turbulent interface in high Reynolds number boundary layers[1] , CHARLES MENEVEAU, Johns Hopkins University, CHARITHA M. DE SILVA, JIMMY PHILIP, KAPIL CHAUHAN, IVAN MARUSIC, University of Melbourne — The scaling and surface area properties of the wrinkled surface separating turbulent from non-turbulent regions in open shear flows are important to our understanding of entrainment mechanisms at the boundaries of turbulent flows. PIV data from high Reynolds number turbulent boundary layers covering three decades in scale are used to resolve the turbulent/non-turbulent interface experimentally and to determine unambiguously that such surfaces exhibit fractal scaling with box-counting exponent between -1.3 and -1.4. A complementary analysis based on spatial filtering of the velocity fields also shows power-law behavior of the coarse-grained interface length as a function of filter width, with an exponent between -0.3 and -0.4. These results establish that the interface is fractal-like with a multiscale geometry and fractal dimension of D $\sim$ 2.3-2.4. Measurements of viscous, subgrid-scale and turbulent fluxes across the interface at various scales confirm the complementary nature of viscous nibbling at small scales while turbulent and then large-scale engulfment dominate when viewed at large scales.

[1]Financial support provided by the Australian Research Council, Fulbright, Melbourne U. and the NSF (CBET 1033942).

1:18PM R31.00002 Geometrical properties and scaling of the turbulent-nonturbulent interface in boundary layers[1] , GUILLEM BORRELL, Universidad Politécnica de Madrid, JAVIER JIMÉNEZ, Universidad Politecnica de Madrid — The turbulent-nonturbulent interface of a boundary layer at $Re_\theta = 1500 - 6500$ is analyzed by thresholding the vorticity magnitude field. The value of the threshold, whithin the range spanning the topological transition from smooth to turbulent, is considered a parameter, and the resulting surface is processed. Its geometrical properties like the relative position to the wall, the fractal dimension or the genus change significantly within the range spanning the topological transition, but the width of the transition scales well with Re_τ when outer units for the vorticity magnitude ($|\omega|^+/\sqrt{\delta_{99}^+}$) are used. The properties of the flow relative to the position of the surface are analyzed within the same range of thresholds, using as a definition of distance the radius of the smallest sphere centerd at the point and touching the surface. That definition works for arbitrarily complex surfaces. The properties of the flow at a given distance to the surface also depend on the threshold, but the average vorticity jump close to the surface scales with the Kolmogorov length, while the average vorticity jump away from the surface scales with the boundary layer thickness.

[1]Funded by ERC, PRACE, CICYT and Spanish Ministry of Economy.

1:31PM R31.00003 Structure of the turbulent/non-turbulent interface of turbulent boundary layers - DNS results , TAKASHI ISHIHARA, Nagoya University, JST CREST, HIROKI OGASAWARA, Nagoya University, JULIAN C. R. HUNT, University College London — Direct numerical simulations (DNS) of turbulent boundary layers (TBL) along a flat plate are used to study the properties of turbulent/non-turbulent (T/NT) interface of the TBL. The values of the momentum-thickness-based Reynolds numbers, Re_θ, used for this study, are $500-2200$. Analysis of the conditional statistics near the interface of the TBL shows that there is a small peak in the span-wise vorticity, and an associated small jump in stream-wise velocity. It is shown that the interfacial layer has a double structure which consists of a turbulent sub-layer with thickness of the order of the Taylor micro scale and its outer boundary (super layer) with thickness of the order of the Kolmogorov length scale. An approximate profile of the conditional average of span-wise vorticity near the interface fits well to the DNS data. The velocity jump near the T/NT interface of the TBL is of the order of the rms value of velocity fluctuations near the interface. Conditional cross correlations of the stream-wise or the wall-normal velocity fluctuations change sharply across the interface, which are consistent with the blocking mechanism of the interface (Hunt and Durbin 1999).

1:44PM R31.00004 A Comparison of the Scalar and Vorticity Criterion defining the T/NT Interface , JONAS BOSCHUNG, FABIAN HENNIG, NORBERT PETERS, Institute for Combustion Technology — Free shear flows are characterized by a turbulent core region, a non-turbulent outer flow and a turbulent/non-turbulent interface separating the two zones. While there exist different approaches to identify this transitional region, the interface position is mostly defined to coincide with the isoscalar surfaces of either a passive scalar or the magnitude of the vorticity. Both criteria are examined and compared using a shear layer DNS.

1:57PM R31.00005 The strain field across the turbulent/non-turbulent interface , GERRIT ELSINGA, Delft University of Technology, RODRIGO TAVEIRA, CARLOS DA SILVA, IST- Technical University of Lisbon — The average flow velocity field associated to the local strain at the turbulent/non-turbulent (T/NT) interface is evaluated using existing Direct Numerical Simulations (DNS) of turbulent planar jets and shear free turbulence. The strain field is of interest as it sets the size of the smallest eddies at the interface (through the dissipation) and governs vorticity stretching, which contributes to the entrainment velocity. Moreover, a similar strain field analysis of the internal turbulence yielded shear layer structures, which appeared universal across different flows and representative of the small-scale features of turbulence. Hence, the principal strain axes provide a meaningful basis for comparing different flows and internal versus interface structure. The results for the T/NT interfaces reveal a shear layer structure separating larger scale motions on either side. The non-turbulent side is characterized by a saddle topology without vorticity, whereas the turbulent side shows a nearly uniform flow parallel to the layer. Thus the larger-scales in the flow not just only determine the interface surface area, but they directly affect the strain and vorticity stretching at the interface and thereby entrainment.

2:10PM R31.00006 Lagrangian evolution of fluid particles in the vicinity of the turbulent non-turbulent interface of a turbulent boundary layer[1] , CALLUM ATKINSON, PAUL STEGEMAN, LTRAC, Monash Univeristy, Australia, JASON HACKL, GUILLEM BORRELL, School of Aeronautics, Universidad Politecnica de Madrid, Spain, JULIO SORIA, LTRAC, Monash Univeristy, Australia — Lagrangian evolution of a fluid in the immediate vicinity of the turbulent non-turbulent interface (TNT) of a turbulent boundary layer ($Re_\tau \approx 800$) is investigated via direct numerical simulation (DNS) and particle tracking. The TNT interface in the DNS is seeded with particles whose evolution in position, velocity and velocity gradient tensor (VGT) are calculated. The velocity and velocity gradients at each particle are determined using a third order Hermite spline interpolation. Probability density functions (PDFs) associated with the change in position, velocity, enstrophy and dissipation are calculated and the evolution of the local flow topology of the fluid near the TNT is examined in terms of the 2nd and 3rd invariants of the VGT (Q and R). Evolution in the QR-plane is compared with conditional mean evolution trajectories and the entrainment and expulsion of particles across the interface are discussed.

[1]Research was initiated as part of the First Multiflow Summer Workshop with support from the European Research Council and is also supported by the Australian Research Council

2:23PM R31.00007 The turbulence boundary of a temporal jet[1] , MARKUS HOLZNER, Institute of Environmental Engineering, ETH Zurich, Zurich, Switzerland, MAARTEN VAN REEUWIJK, Department of Civil and Environmental Engineering, Imperial College London, London, UK — We study the turbulence boundary of temporal plane jet at Reynolds number Re=5000 obtained from a direct numerical simulation. The analysis is based on statistics conditioned on the enstrophy spanning 24 orders of magnitude and identifying essentially irrotational fluid outside the jet to fully turbulent fluid in the jet core. At the jet boundary we find a viscous superlayer (VSL) that envelopes the turbulence. We further identify a turbulent core region (TC) and a buffer region connecting the VSL and the TC. The BR shows many similarities with the turbulent-nonturbulent interface (TNTI), although the TNTI seems to extend into the TC. The BR thickness is about 10 Kolmogorov length scales or half a Taylor length scale, which implies that intense turbulence and viscosity-dominated regions are in close proximity to each other.

[1]The computational resources for this work were provided by the Imperial College HPC facilities. Both authors benefited greatly from an inspiring MULTIFLOW workshop on the Turbulent/Non-turbulent Interface which took place in October 2012 in Madrid.

2:36PM R31.00008 Scaling of the viscous superlayer in zero pressure gradient turbulent boundary layers , KAPIL CHAUHAN, JIMMY PHILIP, IVAN MARUSIC, The University of Melbourne — Scaling of the viscous superlayer (a thin region that exists at the interface of a turbulent boundary layer and a non-turbulent free-stream) is sought using theoretical reasoning and experimental evidence. A kinetic energy criteria is successfully utilised to identify the turbulent/non-turbulent interface over two-dimensional velocity fields in the streamwise/wall-normal plane. The data-analysis utilises particle image velocimetry measurements at four different Reynolds numbers ($\delta^+ = \delta u_\tau/\nu$=1200-14500). The presence of a viscous superlayer is illustrated in all four data sets. It is found that the mean normal velocity across the interface and the tangential velocity jump scales with the skin-friction velocity u_τ. The width of the superlayer is characterised by the local vorticity thickness δ_ω and scales with the viscous length scale ν/u_τ. An order of magnitude analysis of the tangential momentum balance within the superlayer indicates that the turbulent motions also scale with inner scaling, i.e. u_τ and ν/u_τ are the velocity and length scales, respectively.

2:49PM R31.00009 The viscous superlayer in turbulent planar jets , RODRIGO TAVEIRA, CARLOS SILVA, IST-Technical University of Lisbon, CARLOS SILVA TEAM — Direct numerical simulations of turbulent planar jets are used to study the characteristics of the turbulent/non-turbulent interface (TNTI) separating the turbulent from the irrotational regions of the jet, and to define and visualize the viscous super-layer. Conditional statistics near the TNTI show the existence of a region of a region of "dominating enstrophy diffusion and negligible enstrophy production," outside the TNTI. This is the much debated viscous super-layer. The super layer is not continuos and its thickness is of the order of the Kolmogorov micro-scale.

3:02PM R31.00010 Effects of mean shear on the local turbulent entrainment process , MARC WOLF, MARKUS HOLZNER, BEAT LÜTHI, DOMINIK KRUG, WOLFGANG KINZELBACH, Institute of Environmental Engineering, ETH Zurich, 8093 Zurich, Switzerland, ARKADY TSINOBER, School of Mechanical Engineering, Faculty of Engineering, Tel Aviv University, Tel Aviv 69978, Israel — We report on effects of mean shear on the turbulent entrainment process focusing in particular on their relation to small scale processes in the proximity of the turbulent/non-turbulent interface (TNTI). Three-dimensional particle tracking velocimetry measurements of an axisymmetric jet are compared to data from a direct numerical simulation of a zero-mean-shear flow. Conditional statistics relative to the interface position are investigated in a pseudo-Eulerian (i.e. in a fixed frame relative to the interface position) and in a Lagrangian view. A mapping between distance to the instantaneous interface versus conditional time along the trajectory shows that entraining particles remain initially close to the TNTI. Furthermore, decomposing the local entrainment velocity v_n into mean and fluctuating components, we find that mean shear enhances the local entrainment velocity via inviscid and viscous effects.

3:15PM R31.00011 Characteristics of Turbulent/non-turbulent Interface in a Turbulent Planar Jet with a Chemical Reaction[1] , TOMOAKI WATANABE, Nagoya University, JSPS Research Fellow, YASUHIKO SAKAI, KOUJI NAGATA, OSAMU TERASHIMA, YASUMASA ITO, Nagoya University, TOSHIYUKI HAYASE, Tohoku University — Characteristics of chemical reaction (A + B $\rightarrow$ P) near the turbulent/non-turbulent (T/NT) interface are investigated by using the direct numerical simulation of reactive planar jet. The reactants A and B are separately premixed into the jet and ambient flows, respectively. DNS is performed at three different Damköhler numbers. The conditional statistics conditioned on the distance from the T/NT interface is used to investigate the chemical reaction near the T/NT interface. The conditional mean concentration of product P shows a sharp jump near the T/NT interface, and the product P hardly exists in the non-turbulent region. This implies that the chemical reaction takes place in the turbulent region after the reactant B in the ambient flow is entrained into the turbulent region. The conditional mean scalar dissipation rate of mixture fraction has a large peak value slightly inside the T/NT interface. At the same point, the chemical reaction rate also has a peak value in the case of large Damköhler number. On the other hand, when the Damköhler number is small, the chemical reaction rate near the T/NT interface is smaller than that in the turbulent region.

[1]This work was carried out under the Collaborative Research Project of the Institute of Fluid Science, Tohoku University. Part of this work was supported by JSPS KAKENHI Grant Number 25002531 and MEXT KAKENHI Grant Numbers 25289030, 25289031, 2563005.

3:28PM R31.00012 Experimental study of entrainment in a gravity current via a combined Scanning PTV/LIF-technique , DOMINIK KRUG, MARKUS HOLZNER, BEAT LÜTHI, MARC WOLF, WOLFGANG KINZELBACH, ETH Zurich, ARKADY TSINOBER, Tel Aviv University — In this study, we report experimental results on small-scale entrainment characteristics in an inclined dense gravity current. The measurements were performed at $Re = 3700$ and a bulk Richardson number of $Ri = 0.26$. In an investigation domain located at the average interface position between turbulent and non-turbulent flow the instantaneous velocity and density field were obtained through simultaneous Scanning Particle Tracking Velocimetry (PTV) and Scanning Laser Induced Fluorescence (LIF). The data allow us to study the influence of buoyancy on small scale physics governing the turbulent/non-turbulent interface and entrainment, e.g. the contribution of the baroclinic torque to the local entrainment velocity.

Tuesday, November 26, 2013 1:05PM - 3:15PM –

Session R32 Particle-Laden Flows VII: Computational Methods 403 - Eric C. Keaveny, Imperial College of London

1:05PM R32.00001 A computational scheme for simulation of dense suspensions of arbitrarily shaped rigid particles[1] , MARCOS VANELLA, HUSSEIN EZ ELDIN, The George Washington University, PRATEETI MOHAPATRA, CHRISTOPHER DALEY, University of Chicago, ANSHU DUBAY, Lawrence Berkeley National Laboratory, ELIAS BALARAS, The George Washington University — Flows of dense particle suspensions are of great interest to engineering, science and medicine. Immerse boundary (IB) methods are commonly employed in simulations of such systems, but are usually confined to spherical particle suspensions. Extension to rigid particles of arbitrary shape introduces significant additional complexities on the IB tracking algorithms, as well as the rigid body dynamics. This increases the cost in the fluid-structure interaction (FSI) schemes employed. In this work we present a computational scheme targeted to the above problem, applicable to computations involving millions of particles on leadership high performance computing platforms. The fluid equations are discretized by standard, central, finite-differences on a staggered mesh and the equations of motion for each particle are employed on the Eulerian reference using Euler angles or quaternion variables. A Lagrangian forcing IB method is employed, using the Lagrangian particle framework of FLASH. Fluid and particle equations of motion are strongly coupled using a partitioned scheme. We present the details of the parallel implementation as well as scaling tests and results on the sedimentation of arbitrary shaped particles.

[1]Work supported by NSF (OCI-0904920) and utilized XSEDE (TG-CTS110028)

1:18PM R32.00002 An investigation of particles suspension using smoothed particle hydrodynamics[1] , ARMAN PAZOUKI, DAN NEGRUT, University of Wisconsin-Madison — This contribution outlines a method for the direct numerical simulation of rigid body suspensions in a Lagrangian-Lagrangian framework using extended Smoothed Particle Hydrodynamics (XSPH) method. The dynamics of the arbitrarily shaped rigid bodies is fully resolved via Boundary Condition Enforcing (BCE) markers and updated according to the general Newton-Euler equations of motion. The simulation tool, refered to herien as *Chrono::Fluid*, relies on a parallel implementation that runs on Graphics Processing Unit (GPU) cards. The simulation results obtained for transient Poiseuille flow, migration of cylinder and sphere in Poiseuille flow, and distribution of particles at different cross sections of the laminar flow of dilute suspension were respectively within 0.1%, 1%, and 5% confidence interval of analytical and experimental results reported in the literature. It was shown that at low Reynolds number, Re = O(1), the radial migration (a) behaves non-monotonically as the particles relative distance (distance over diameter) increases from zero to two; and (b) decreases as the particle skewness and size increases. The scaling of *Chrono::Fluid* was demonstrated in conjunction with a suspension dynamics analysis in which the number of ellipsoids went up to 3e4.

[1]Financial support was provided in part by National Science Foundation grant NSF CMMI-084044.

1:31PM R32.00003 Numerical simulation of two-way coupling mechanism in particle-laden turbulent flow based on one-dimensional turbulence model , GUANGYUAN SUN, DAVID LIGNELL, Brigham Young University, JOHN HEWSON, CRAIG GIN, Sandia National Laboratories — We present three algorithms (type-I, type-C and type-IC) for Lagrangian particle transport within the context of the one-dimensional turbulence (ODT) approach. ODT is a stochastic model that captures the full range of length and time scales and provides statistical information on fine-scale turbulent-particle mixing and transport at low computational cost. Two of the particle transport algorithms are new as is an algorithm to provide two-way momentum and energy coupling between the particle and carrier phases. Using these methods we investigate particle-laden turbulent jet flow. In contrast to other previous particle implementation in ODT, the two new methods allow the particles to interact with multiple eddies simultaneously and evolve the particle phase continuously, and therefore are able to accurately capture turbulent mixing and fluctuation seen by inertial particles in ODT. Simulation results are compared with experimental data including the effect of two particle Stokes numbers (St = 3.6 and 10.8). Turbulence modification, particle number density PDFs and particle velocity evolution are presented.

1:44PM R32.00004 Incorporating Volumetric Displacement Effects In Euler-Lagrange Simulations of Particle-Laden Oscillatory Flows[1] , SOURABH APTE, Oregon State University, JUSTIN FINN, University of Liverpool, ANDREW CIHONSKI, Los Alamos National Laboratory — Recent Euler-Lagrange discrete element modeling of a few microbubbles entrained in a traveling vortex ring (Cihonski et al., JFM, 2013) has shown that extension of the point-particle method to include local volume displacement effects is critical for capturing vortex distortion effects due to microbubbles, even in a very dilute suspension. We extend this approach to investigate particle-laden oscillatory boundary layers representative of coastal sediment environments. A wall bounded, doubly periodic domain is considered laden with a layer of sediment particles in laminar as well as turbulent oscillatory boundary layers corresponding to the experiments of Keiller and Sleath (1987) and Jensen et al. (1987). Inter-particle and particle-wall collisions are modeled using a soft-sphere model which uses a nested collision grid to minimize computational effort. The effects of fluid mass displaced by the particles on the flow statistics are quantified by comparing a standard two-way coupling approach (without volume displacement effects) with volume displacement effects to show that the latter models are important for low cases with low particle-fluid density ratios.

[1]NSF project #1133363, Sediment-Bed-Turbulence Coupling in Oscillatory Flows. EPSRC Project # EP/J00507X/1, EP/J005541/1 Sand Transport under Irregular and Breaking Waves Conditions (SINBAD)

1:57PM R32.00005 Fluctuating force-coupling method for simulating Brownian suspensions , ERIC KEAVENY, Imperial College London — Brownian motion plays an important role in the dynamics of colloidal suspensions. It affects suspension rheological properties, influences the self-assembly of structures, and regulates particle transport. While including Brownian motion in simulations is necessary to reproduce and study these effects, it is computationally intensive due to the configuration dependent statistics of the particles' random motion. I will discuss recent work that speeds up this calculation for the force-coupling method (FCM), a regularized multipole approach to simulating suspensions at large-scale. I will show that by forcing the surrounding fluid with a configuration independent, white-noise stress, fluctuating FCM yields the correct particle random motion, even when higher-order terms, such as the stresslets, are included in the multipole expansion. I will present results from several simulations demonstrating the effectiveness of this approach and also discuss the extension of fluctuating FCM to dense suspension simulations.

2:10PM R32.00006 Particle-Laden Turbulent Kolmogorov Flow[1] , LIAN-PING WANG, University of Delaware — Modulation of the carrier phase turbulence by finite-size inertial particles have been studied experimentally, but only recently it is possible to study this computationally through particle-resolved simulation methods. In addition to parameters governing the flow, the nature of modulation depends on at least four dimensionless parameters associated with the dispersed phase: the dimensionless particle size, volume fraction, particle-to-fluid density ratio, and dimensionless sedimentation velocity. Both augmentation and attenuation of the carrier phase turbulence have been reported, and the published results are often difficult to comprehend and sometime are inconsistent. Here we present results of a relatively simple setting, namely, a turbulent Kolmogorov flow laden with finite-size inertial particles. This flow setting has connection to both channel flow and homogeneous flow. We apply the lattice Boltzmann method to simulate the carrier phase turbulence and to resolve the surface of moving solid particles. Both turbulent augmentation and attenuation are found to exist, depending on the system parameters. We will report on results of large-scale energy production and local profiles near the particle surface, to help interpret the results of turbulence modulation.

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

2:23PM R32.00007 Study of the Motion of Particles in Closed Streamlines[1] , HAMED HADDADI, KEVIN CONNINGTON, Levich Institute, City College of New York, SHAHAB SHOJAEI-ZADEH, Rutgers University, JEFFREY MORRIS, Levich Institute, City College of New York — The behavior of neutrally-buoyant particles in the closed-streamline flows formed behind bluff bodies of various shapes is studied; the Reynolds numbers studied generate extended closed-streamline wakes but are below the transition to an unsteady wake. Experimental observations have demonstrated that the wake is depleted or completely devoid of particles. Using lattice-Boltzmann simulations, the trajectory of a single particle (small relative to the bluff body) is analyzed and shown to form a limit cycle inside the wake. With increase of the number of particles in the wake, trajectories are distorted due to interactions and particles are pushed out of the wake. Calculation of the fluid pathlines indicates that the presence of particles breaks the steadiness of the wake which results in a particle (and fluid) transfer between the wake and the free stream. The particle trajectories have also been analyzed by simulation of the flow of dilute suspensions over the circular cylindrical, square and thin rectangular ("blade") shaped posts, for which different levels of particle depletion in the wake are seen experimentally, in order to determine the particle transfer pattern between the wake and the free stream.

[1]NSF 0853720

2:36PM R32.00008 A New Moving Boundary Condition in Particulate Suspensions with the Lattice Boltzmann Method , LINA XU, LAURA SCHAEFER, University of Pittsburgh — Particulate suspensions are common phenomena in industrial and biological fields. However, the fundamental understanding of the hydrodynamic interactions between the solid and fluid needs to be further improved. The lattice Boltzmann method has been shown to be an effective numerical method to model various fluid flows, and exhibits good performance in dealing with boundary conditions, with straightforward and easy-to-implement methods for complex solid boundaries. However, most of the previous boundary conditions used for the moving complex surface are based on the half way bounce-back boundary condition, where the geometric integrity of the body cannot be kept. In this presentation, a new boundary condition based on the Chapman-Enskog expansion is proposed for the moving complex surface, where the precise shape of the body can be preserved during the calculation. Moreover, due to the second order accuracy of the Chapman-Enskog expansion when recovering the Navier-stokes equation from the Boltzmann-BGK equation, the new boundary condition can maintain the same accuracy for the whole computational domain.

2:49PM R32.00009 Lattice Boltzmann Method for Two-phase Flows on Unstructured Mesh[1] , TAEHUN LEE, LINA BAROUDI, City College of City University of New York, KENT WARDLE, Argonne National Laboratory — A lattice Boltzmann method with Galerkin finite element discretization (FE-LBM) is proposed to simulate incompressible two-phase flows on unstructured mesh. Two-distribution functions are used to recover the transport equations for the order parameter, pressure, and momentum. Consistent treatment of streaming and intermolecular forcing terms in FE-LBM enables us to use small equilibrium interface thickness compared with the existing two-phase LBMs and thus to achieve numerical stability at higher Reynolds number and large material property contrast. Several benchmark test cases with non-trivial wall boundaries will be presented, which include turbulent free surface flow inside a concentric rotating cylinder, drop impact on patterned surfaces, and bubbly flows.

[1]This work is partially supported by the DOE's NEUP.

3:02PM R32.00010 An improved lattice Boltzmann method for incompressible two-phase flows with large density differences , TAKAJI INAMURO, TAKAAKI YOKOYAMA, KENTARO TANAKA, MOTOKI TANIGUCHI, Dept. Aeronautics and Astronautics, Kyoto University — We propose a new LBM for two-phase fluid flows with high density ratios by improving the pressure computing of Inamuro et al.'s method (2004) [J. Comput. Phys. 198 (2004) 628] without solving the pressure Poisson equation. In the proposed method, the velocity and pressure fields are computed by using a single velocity distribution function even for high density ratios and by adjusting the speed of sound in a high density region to satisfy the continuity equation. In order to show the validity of the method, we apply the method to the simulations of a stationary drop, binary droplet collision, rising bubbles, and a milk crown. In a stationary drop, pressure and density profiles are computed, and the effect of a sound speed on time evolution of the pressure field in the drop. In the simulations of a binary droplet collision and rising bubbles, the computed results by the proposed method are compared with those by Inamuro et al.'s method (2004). A thin sheet and tiny drops can be computed in the simulation of a milk crown.

Tuesday, November 26, 2013 1:05PM - 3:15PM –

Session R33 Drops XV: Superhydrophobic Surfaces 404 - Xin Yong, University of Pittsburgh

1:05PM R33.00001 Slide, Sweep and Vanish: Droplet manipulation by wettability engineering , ARITRA GHOSH, RANJAN GANGULY[1], THOMAS M. SCHUTZIUS, CONSTANTINE M. MEGARIDIS, Mechanical and Industrial Engineering, University of Illinois at Chicago — Achieving controlled droplet transport on substrates is important for multiphase heat transfer, water harvesting and lab-on-chip applications. We use a facile, scalable surface wettability engineering approach to generate wettability patterned surfaces that comprise of superhydrophilic tracks of various geometrical patterns and length scales (μm-mm) on superhydrophobic backgrounds. Liquid transport on such surfaces harnesses the force arising from the spatial contrast of surface energy on the substrate, providing rapid actuation for micro and nanoliter drops. Considering a variety of dimensions, shapes and strategic locations of the superhydrophilic patterns on the substrate, effective modes of droplet transport through hemiwicking and Laplace pressure-driven flow are analyzed. The work provides proof-of-concept for salient digital microfluidic tasks, e.g. droplet capture, transport, merging and dispensing on such patterned substrates. This droplet manipulation is pumpless and fast. With suitable patterns and wettability contrast, we demonstrate on-chip droplet transport speeds of O(10 cm/s). The study examines the geometric and surface wettability parameters for optimal substrate design for droplet manipulation.

[1]On leave from Jadavpur University, India

1:18PM R33.00002 Dynamic Wetting of a Droplet on a Hydrophobic Micro-patterned Surface , XIAN WEI, HUAN LI, CHENG WANG, XIN TANG, SASCHA HILGENFELDT, K. JIMMY HSIA, Mechanical Science and Engineering, University of Illinois at Urbana-Champaign — The dynamics of droplets moving over patterned surfaces of micro-pillar arrays is of great practical interest, but has lacked detailed study at the level of the micron-scale pattern. We develop an imaging method and a force measurement setup to study contact line (CL) evolution and contact angle hysteresis (CAH) induced resistant force for a water droplet sliding on PDMS micro-pillar arrays. The topography of the CL between droplet and surface is imaged using fluorescence microscopy in combination with high-speed video. To measure the CAH induced resistant force, a micro-force sensor is attached to the droplet and the substrate moved relative to the droplet with prescribed velocity. The resultant force-time curve displays an initial maximum and subsequently a dynamic steady state with a sawtooth-like shape. The ratio between the average force at maximum and that in the dynamic balance state is approximately constant and close to the ratio between pillar numbers at the trailing edge at both states. In steady state, the shape of the force curve can be correlated with events of pillar pinning and depinning at the CL to obtain a detailed understanding of attachment and detachment forces.

1:31PM R33.00003 Simulations of Droplets on Micro-patterned Surfaces , BANGLIN LIU, MICHAEL GRIGOLA, HUAN LI, SASCHA HILGENFELDT, K. JIMMY HSIA, Mechanical Science and Engineering, University of Illinois at Urbana-Champaign — The behavior of liquid droplets on micro-patterned surfaces made from arrays of micropillars is important for applications in self-cleaning surfaces, refrigeration, or pore filtration. Properties like droplet contact angles and their hysteresis have been described in macroscopic terms from coarse-grained variables like pillar density. However, for accurate modeling of the droplet shape and dynamical behavior, microscopic parameters like pillar positioning and the topography of the contact line are crucial. We have developed an energy-based model of a water droplet on a PDMS substrate in the Cassie-Baxter state using Surface Evolver. We assess the changes in droplet energy upon deformation and displacement, with particular attention to the pinning and depinning from individual pillars. The majority of shape distortion and energy change is found to occur in close proximity to the substrate, encouraging a simplified theoretical description using concepts of 2D contact-line pinning. The versatile simulation tool can be used to study the effects of pillar position pattern, pillar orientation, substrate symmetry, and many more general problems of contact-line statics and dynamics.

1:44PM R33.00004 Effect of Vapor Flow on Jumping Droplets during Condensation on Superhydrophobic Surfaces , DANIEL J. PRESTON, NENAD MILJKOVIC, Massachusetts Institute of Technology, RYAN ENRIGHT, Bell Labs Ireland, ALEXANDER LIMIA, EVELYN N. WANG, Massachusetts Institute of Technology — Upon coalescence of droplets on a superhydrophobic surface, the net reduction in droplet surface area results in a release of surface energy that can cause the coalesced droplet to "jump" away from the surface. Jumping condensing surfaces have been shown to enhance condensation heat transfer by up to 30% compared to state-of-the-art dropwise condensing surfaces. While the heat transfer enhancement of jumping condensation is well documented, droplet behavior after departure from the surface has not been considered. Vapor flows to the condensing surface due to mass conservation. This flow can increase drag on departing droplets, resulting in complete droplet reversal and return to the surface. Upon return, these larger droplets impede heat transfer until they jump again or finally shed due to gravity. By characterizing individual droplet trajectories during condensation on hydrophobic nanostructured copper oxide surfaces for a variety of heat fluxes (q" $= 0.1 - 2$ W/cm2), we showed that vapor flow entrainment dominates droplet motion for droplets smaller than $R \approx 30$ um at high heat fluxes (q" >2 W/cm2). Furthermore, we developed an analytical model of droplet motion based on first principles and the Reynolds drag equation which agreed well with the experimental data. We considered condensation on both flat and tubular geometries with our model, and we suggest avenues to further enhance heat transfer which minimize droplet return due to entrainment.

1:57PM R33.00005 Cold-induced Spreading of Water Drops on Hydrophobic Surfaces , FARYAR TAVAKOLI, PIROUZ KAVEHPOUR, UCLA — Superhydrophobic surfaces received tremendous attention in recent years mainly due to their self–cleaning properties. Wenzel and Cassie–Baxter models for relating stable equilibrium contact angle to physical parameters of liquid and solid ignore tangible factors such as temperature and humidity. Here, we show a peculiar behavior of equilibrium contact angle on cold hydrophobic surfaces. Water drops were cooled by a peltier element to temperatures below the melting point of water and, surprisingly, substantial change in static contact angle and base diameter were observed during the cooling process. Physical variables such as substrate temperature, humidity, drop volume, and even fabrication type of hydrophobic surfaces are found to be detrimental to post–spreading shape.

2:10PM R33.00006 Direct observation of self-similar contact line depinning from superhydrophobic surfaces , ADAM PAXSON, KRIPA VARANASI, Massachusetts Institute of Tehcnology — The adhesion of a drop to a superhydrophobic surface, although very low, is never altogether eliminated. As the drop moves along the surface, the advancing portion of the contact line simply lies down onto the upcoming roughness features, contributing negligibly to adhesion. Instead, the pinning and contact angle hysteresis are governed by the depinning of capillary bridges formed at the receding portion of the contact line. We use environmental scanning electron microscopy to observe these depinning events at the microscale. After measuring the local receding contact angle of capillary bridges formed on a micropillar array, we find that these depinning events follow the Gibbs depinning criterion. We further extend this technique to two-scale hierarchical structures to reveal a self-similar depinning mechanism in which the adhesion of the entire drop depends only on the pinning at the very smallest level of roughness hierarchy. With this self-similar depinning mechanism we develop a model to predict the adhesion of drops to superhydrophobic surfaces that explains both the low adhesion on sparsely structured surfaces and the surprisingly high adhesion on surfaces whose features are densely spaced or tortuously shaped.

2:23PM R33.00007 Technique for needle-free drop deposition: Pathway for precise characterization of superhydrophobic surfaces , PRASHANT R. WAGHMARE, SIDDHARTHA DAS, SUSHANTA K. MITRA, Department of Mechanical Engineering, University of Alberta, Edmonton, Alberta, Canada T6G 2G8 — The most important step for characterizing the wettability of a surface is to deposit a water drop on the surface and measure the contact angle made by the drop on the surface. This innocuously simple process relies on bringing a needle holding the water drop in close proximity to the surface, with a "desire" that the drop would spontaneously detach from the needle and get deposited on the surface. Problem occurs when the surface is superhydrophobic, expressing an "unwillingness" to "see" the water drop in preference to a much more "water-loving" needle surface. There exists no solution to this problem, and surfaces are invariably characterized where the drop-needle assembly contacts the superhydrophobic surface. Such a configuration will always lead to an incorrect estimation of the contact angle, as there is no certainty of the existence of the drop-surface contact. Here we shall discuss our recently invented technique, where we solve this long-standing problem–we indeed ensure a needle-free drop in contact with the superhydrophobic surface, thereby ascertaining precise determination of the contact angle. The successful application of the technique will address a major headache of the big research community interested in science and technology of superhydrophobic surfaces.

2:36PM R33.00008 New drop deposition technique for wettability characterization of under-liquid superoleophobic surfaces , SUSHANTA MITRA, PRASHANT WAGHMARE, SIDDHARTHA DAS, University of Alberta — From understanding the remarkable self-cleaning behavior of fish scales to the preparation of surfaces that will counter the destructive effects of oil-spills, there has been a remarkable interest in understanding the wettability of a solid in an "under-liquid" configuration. Like surfaces in air, here too, the main focus remain in designing surfaces (such as fish scales) that exhibit repelling behavior to a multiple other liquids in this "under-liquid" state. Problem occurs, just as with surfaces in air, when this "under-liquid" surface is too repelling to a given liquid. In that case, the standard drop deposition technique is unable to deposit a drop that is not "interfered" by the needle holding the drop. Here we shall discuss a unique technique that ensures that we achieve a "needle-free" deposited drop on the under-liquid surface. A drop is produced at the end of the needle, with the needle placed inside the liquid bath. Then the needle holding the drop is moved away from the concerned surface, and the moment this drop-needle assembly hits the liquid-air or liquid-another-liquid (a layer of this another liquid is intentionally created at the location where the liquid bath is exhausted), the surface tension effects will ensure that the drop is detached from the needle.

2:49PM R33.00009 Liquid transfer between two solid surfaces with the effect of contact angle hysteresis , HUANCHEN CHEN, TIAN TANG, Department of Mechanical Engineering, University of Alberta, ALIDAD AMIRFAZLI, Department of Mechanical Engineering, York University — Drop transfer from one solid surface to another (e.g. due to the approach of a surface from top to a sessile drop resting on a lower surface) is widely observed in many industrial areas, e.g. offset printing. This process is governed by many factors such as the contact angle (CA) and contact angle hysteresis (CAH) of surfaces, viscosity of the liquid and the rate at which the donor and acceptor surfaces are separated. In this work, an experimental apparatus is developed to study the transfer of liquid drop between surfaces, with the particular focus on addressing the effect of the surfaces' CAH when the loading speed is low (transfer is quasi-static). In the experiment, a liquid bridge between the two surfaces is first formed by compression; then stretched to the point of breakage. By using surfaces that have similar CA but dissimilar CAH, the liquid transfer ratio (the amount of liquid transferred to the acceptor surface over the total amount of liquid) is found to be significantly influenced by CAH. In addition, as a result of CAH, the maximum compression of the liquid bridge is found to play an important role in determining the transfer ratio. These findings can be very helpful for the design of surfaces and loading conditions to achieve desired transfer ratios in practice.

3:02PM R33.00010 Passing of a drop from one surface to another: a simulation and analytical study , ALIDAD AMIRFAZLI, Department of Mechanical Engineering, York University, TIAN TANG, HUANCHEN CHEN, Department of Mechanical Engineering, University of Alberta — Due to its high throughput and cost-effectiveness, offset printing is a widely used printing technique in which the ink is transferred from a substrate to the target surface. A thorough understanding of liquid transfer between two solid surfaces can significantly improve the working efficiency of offset printing. Depending on the approaching and separation speeds of the surfaces, their wettability and liquid viscosity, the liquid transfer can be categorized into two regimes: quasi-static regime where the surface tension force dominates, and dynamic regime where contributions from viscous and inertia forces are not negligible; however, the delineating conditions for these two regimes are not understood. In this work, a numerical model based on the volume of fluid method was developed to study the liquid transfer under different approaching and separation speeds. The effect of dynamic contact angle is included. With this model, the liquid transfer ratio (the amount of liquid transferred to the acceptor surface over the total amount of liquid) is calculated and used to determine the boundary between the quasi-static and the dynamic regimes. A systematic study is conducted on how the transfer ratio is affected by the speed of the surfaces, viscosity of liquid and surface's wettability in the dynamic regime.

Tuesday, November 26, 2013 1:05PM - 3:28PM –

Session R34 Instability: Boundary Layers II - Geometry and Flow Conditions 405 - Jose Eduardo Wesfreid, PMMH

1:05PM R34.00001 Experimental Investigation of Effect of Wall Suction on Cross-Flow Absolute Instability in a Rotating Disk Boundary Layer , JOANNA HO, THOMAS CORKE, ERIC MATLIS, University of Notre Dame — The research is intended to investigate the effect of uniform suction on the absolute instability of Type I cross-flow modes on a rotating disk. Specifically it is designed to investigate if wall suction will transform the absolute instability into a global mode as postulated in the numerical simulations of Davies and Carpenter (2003). The disk is designed so that with a suction parameter of $a = w/(\nu\omega)^{1/2} = 0.4$, the radial location of the absolute instability critical Reynolds number, $R_{C_a} = 803$, occurs on the disk. Uniform wall suction is applied from $R = 449$ to 919. The design for wall suction follows that of Gregory and Walker (1950), where an array of holes through the disk communicate between the measurement side of the disk and the underside of the disk in an enclosure that is maintained at a slight vacuum. The holes in the measurement surface are covered by a stretched silk cloth that provides a smooth, finely porous surface. A companion numerical simulation was performed to investigate the effect of the size and vacuum pressure of the underside enclosure on the uniformity of the measurement surface suction. Temporal disturbances are introduced using the method of Othman and Corke (2006), and the evolution of the resulting wave packets is documented.

1:18PM R34.00002 The flow along an external corner revisited , JIM DENIER, The University of Auckland, NATHANIEL JEWELL, The University of Adelaide — We revisit the problem of the flow of an almost inviscid fluid along an external corner made from the junction of two quarter infinite plates joined at an angle $0 < \alpha < \pi/2$. The structure of the boundary layer which develops along the corner is explored using a computational approach based upon a spectral element discretisation of the steady two-dimensional boundary-layer equations. We pay particular attention to the case when the angle α is small, thus approximating the semi-infinte quarter plate problem considered by Stewartson (1961) and recently revisited by Duck & Hewitt (2012). Our results, which demonstrate a thickening of the boundary-layer near the sharp corner, will be discussed in the context of the asymptotic theory developed in the aforementioned papers.

1:31PM R34.00003 Stabilization by shape optimization , CHRISTOPHE HENNEKINNE, MATTHEW P. JUNIPER, University of Cambridge, Department of Engineering — In a wide range of flows called oscillator flows, the transition to turbulence starts with a modal instability. This first instability can be accurately predicted by performing a linear stability analysis. With the aim of preventing this instability, we examine one of the simplest passive control strategies : the modification of the shape of the boundary. We present a gradient-based algorithm to recover a locally optimal shape of our problem. This algorithm is similar to existing shape optimization algorithms in that it requires computation of the shape gradient, which is the gradient of the objective function with respect to a modification of the boundary. However it differs from existing shape optimization algorithms in the sense that the objective to minimize is not a functional of the flow field but the growth of the most unstable mode of the linearized operator. Consequently two adjoint equations need to be solved sequentially to recover the shape gradient: one associated with the eigenproblem and the other with the steady Navier–Stokes equation. The algorithm is tested on the flow over a backward facing slope. By changing the shape of the slope, the three-dimensional instability that grows on top of the two-dimensional flow is delayed.

1:44PM R34.00004 Instability of the 2-D bottom boundary layer under a solitary wave , MAHMOUD SADEK, PHILIP LIU, Cornell University, LUIS PARRAS, University of Malaga, PETER DIAMESSIS, Cornell University — Fully nonlinear 2-D simulations are used to investigate the temporal instability of the bottom boundary layer (BBL) driven by a soliton-like pressure gradient in an oscillating water tunnel (an approximation of the BBL under a solitary wave). As a function of the associated Reynolds number (Re), the base flow (BF) can be classified as unconditionally stable, conditionally unstable or unconditionally unstable. In the third regime, the BBL flow is unstable, regardless of perturbation amplitude. Two distinct unstable modes emerge in this last regime depending on the value of Re. In the unconditionally unstable regime, we identify the limiting Re value above which instability is observed in the acceleration phase of the BF. The BF profile in this phase lacks a deflection point, suggesting that the above instability is of viscous origin. A sensitivity analysis has been carried out to assess the effect of the different initial perturbation characteristics (i.e. amplitude, spectral shape, time of insertion, e.t.c.) and a variety of wave shapes on the BBL's instability properties for different Re values. In parallel with the fully non-linear simulations, the applicability of both modal and non-modal instability analysis is also examined.

1:57PM R34.00005 New boundary layer structures due to wall slippage , HSIEN-HUNG WEI, National Cheng Kung University, Department of Chemical Engineering — We demonstrate that wall slip can significantly modify temporal and spatial structures of boundary layer flows. Two benchmark problems for flow generated by a moving plate are re-investigated to reveal how the boundary layer thickness δ and the slip length λ determine flow characteristics: (i) Stokes's first problem, and (ii) Blasius's problem. In (i), the solution is found to combine the features of two problems: (a) simple vorticity diffusion driven by a constant wall stress created by strong wall slippage, and (b) the classical Stokes first problem driven by a no-slip moving plate, characterizing short time and long time solution behaviors, respectively. A similar slip-to-no-slip transition can occur *spatially* to (ii), leading the friction law to change from the well-known Blasius law $C_\mathrm{f} \sim Re^{-1/2}$ to the free-surface-like result $C_\mathrm{f} \sim (L/\lambda)Re^{-1}$ when the Reynolds number Re (based on the plate length L) is greater than $(L/\lambda)^2$.

2:10PM R34.00006 Global stability analysis of axisymmetric boundary layers , RAMESHKUMAR BHORANIYA, VINOD NARAYANAN, Mechanical Engineering, Indian Institute of Technology, Gandhinagar, India — Global stability analysis of the axisymmetric boundary layer flow explores the stability features of a nonparallel flow. Consider an incompressible flow past a cylinder, where flow direction is parallel to the axis of cylinder. The ensuing boundary layer is axisymmetric but non-similar. Due to the boundary layer growth, the velocity profile is two dimensional. This work aims to understand the nonparallel effects of an axisymmetric boundary layer using a biglobal stability analysis. The linearized biglobal stability equations are derived in polar cylindrical coordinates. The resulting stability equations along with boundary conditions form an eigenvalue problem, which is solved using Chebyshev spectral collocation method. Arnoldi's algorithm is used to compute selective eigenvalues and eigenfunctions. The results show that the nonparallel effects are considerable at very moderate Reynolds numbers. More detailed results will be presented at the time of conference.

2:23PM R34.00007 Stability of high-speed boundary layers in oxygen including chemical non-equilibrium effects[1] , JILL KLENTZMAN, ANATOLI TUMIN, University of Arizona — The stability of high-speed boundary layers in chemical non-equilibrium is examined. A parametric study varying the edge temperature and the wall conditions is conducted for boundary layers in oxygen. The edge Mach number and enthalpy ranges considered are relevant to the flight conditions of reusable hypersonic cruise vehicles. Both viscous and inviscid stability formulations are used and the results compared to gain insight into the effects of viscosity and thermal conductivity on the stability. It is found that viscous effects have a strong impact on the temperature and mass fraction perturbations in the critical layer and in the viscous sublayer near the wall. Outside of these areas, the perturbations closely match in the viscous and inviscid models. The impact of chemical non-equilibrium on the stability is investigated by analyzing the effects of the chemical source term in the stability equations. The chemical source term is found to influence the growth rate of the second Mack mode instability but not have much of an effect on the mass fraction eigenfunction for the flow parameters considered.

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

2:36PM R34.00008 The influence of the pressure gradient on the development of Görtler vortices , JOSUEL ROGENSKI, LEANDRO F. DE SOUZA, Universidade de São Paulo, Brasil, JERZY M. FLORYAN, University of Western Ontario — The optimization in the process of turbomachinary design demands the ability to predict the transition region. The flow over the concave part of a turbine blade is subjected to centrifugal instability and pressure gradient where streamwise vortices can be formed. These vortices cause strong distortions in the streamwise velocity profile. In this sense, a study of the pressure gradient effect focused on the Görtler vortices development is necessary. The Navier-Stokes equations in the vorticity-velocity formulation are used. It is assumed periodicity in the spanwise direction. A mesh stretching in the normal direction is adopted. The use of Direct Numerical Simulation is necessary to ensure that all relevant scales are correctly be represented. Compact high-order finite difference approximations are adopted in the streamwise and wall normal directions. The temporal advance is done by the classical 4th order Runge-Kutta method. The elliptic problem is solved by the use of a multigrid method. The code is parallelized using a domain decomposition technique. The results indicate that the numerical code is able to simulate the physical phenomena under investigation. The presence of a favorable pressure gradient tends to stabilize the flow.

2:49PM R34.00009 Heat transfer enhancement by the Goertler vortices developed on a wall with a finite thermal conductivity[1] , INNOCENT MUTABAZI, HARUNORI YOSHIKAWA, JORGE PEIXINHO, LYES KAHOUADJI, LOMC, UMR6294, CNRS-Université du Havre — Görtler vortices appear in a flow over a concave wall as a result of centrifugal instability [Saric, Annu. Rev. Fluid Mech. 26, 379 (1994)]. They may have a strong influence on heat transfer [Momayez et al., Int. J. heat Mass transfer 47, 3783(2004)]. The purpose of this work is to model heat transfer by Görtler vortices using a weakly nonlinear analysis of Smith &-Haj- Hariri [Phys. Fluids A5, 2815(1993)]. We have investigated the coupling of the convective heat transfer by the stationary vortices with the heat conduction inside the solid wall. The finite thickness and thermal conductivity of the wall enter into the boundary conditions of the problem through the ratio δ of the wall thickness to the boundary layer thickness and through the ratio K of the thermal conductivities of the fluid and the wall. The parametric dependence $Nu(\delta, K)$ of the Nusselt number is performed and it is shown that found the heat transfer is quite well modified by these two parameters. The local thermal stress can be estimated in order to analyze the effects on ageing of the wall material.

[1]The authors acknowledge the financial support of the french Agence Nationale de la Recherche (ANR), through the program "Investissements d'Avenir" (ANR-10-LABX-09-01), LabEx EMC3.

3:02PM R34.00010 Optimal divergence-free inflow perturbations in flow over an airfoil[1] , SEAN LOH, HUGH BLACKBURN, Monash University, XUERUI MAO, University of Durham — Linear transient growth analysis has identified various key mechanisms in transition due to free-stream turbulence in canonical flow open flow configurations (Durbin & Wu 2007). In the present work, the role of inflow disturbances in promoting transition for flow over airfoil type geometries is examined. Using an optimal control based methodology, optimal divergence-free inflow perturbations for linear transient energy growth are computed for a NACA 0012 airfoil at 4° angle of attack. At various low-to-moderate Reynolds numbers, the flow response to optimal two-dimensional inflow perturbations with varying streamwise length scale is analysed. The relationship between the flow physics induced by optimal inflow perturbations, optimal initial perturbations and leading linear instability modes is then examined. Durbin P & Wu X (2007), Transition beneath vortical disturbances, *Annu. Rev. Fluid Mech.* **39**: 107.

[1]Supported by Australian Research Council grant DP1094851.

3:15PM R34.00011 A 2D pendulum submitted to an incoming flow: drag acting like gravity and new instabilities , ANDREA FANI, Laboratoire Dieudonne, Universite de Nice Sophia-Antipolis, Nice, France and LFMI, EPFL, Lausanne, Switzerland, FRANCOIS GALLAIRE, LFMI, EPFL, Lausanne, Switzerland — Flow induced oscillations of slender bodies facing an incoming flow are relevant in a large number of engineering applications, such as the design of tubular structures of offshore platforms, heat exchangers and energy harvesting. Numerical simulations and experiments available in literature often consider a circular cylinder in an uniform flow which can move only transversally with respect to the flow direction. In a recent work Semin *et al.* (*JFM*, 2011) studied a tethered 2D cylinder strongly confined between two parallel plane walls. It is shown that confinement alters significantly the flow dynamics, with a new instability, denoted confinement induced vibration (CIV), which occur at a Reynolds number much lower than the vortex induced vibration (VIV) critical one. In the present work we characterize the instability scenario of a confined tethered cylinder by means of a global stability analysis of the fluid-structure problem. In strongly confined channels, a periodic unstable mode, related to CIV vibrations, is observed, while for moderated confinement a new steady diverging instability is founded.

Tuesday, November 26, 2013 1:05PM - 2:36PM –

Session R35 Education and Career Outreach: Teaching Methods 406 - Shanon Reckinger, Fairfield University

1:05PM R35.00001 Flippin' Fluid Mechanics – Using Online Technology to Enhance the In-Class Learning Experience , D.R. WEBSTER, D.M. MAJERICH, Georgia Tech — This study provides an empirical analysis of using online technologies and team problem solving sessions to shift an undergraduate fluid mechanics course from a traditional lecture format to a collaborative learning environment. Students were from two consecutive semesters of the same course taught by the same professor. One group used online technologies and solved problems in class and the other did not. Out of class, the treatment group watched 72 short (11 minutes, average) video lectures covering course topics and example problems being solved. Three times a week students worked in teams of two to solve problems on desktop whiteboard tablets while the instructor and graduate assistants provided "just-in-time" tutoring. The number of team problems assigned during the semester exceeded 100. Weekly online homework was assigned to reinforce topics. The WileyPlus online system generated unique problem parameters for each student. The control group received three-50 minute weekly lectures. Data include three midterms and a final exam. Regression results indicate that controlling for all of the entered variables, for every one more problem solving session the student attended, the final grade was raised by 0.327 points. Thus, if a student participated in all 25 of the team problem solving sessions, the final grade would have been 8.2 points higher, a difference of nearly a grade. Using online technologies and teamwork appeared to result in improved achievement, but more research is needed to support these findings.

1:18PM R35.00002 Teaching an Undergraduate Course on Computational Fluid Dynamics , REZA H. SHEIKHI, Northeastern University — A new computational fluid dynamics (CFD) course is introduced to Mechanical Engineering undergraduate curriculum at Northeastern University. The main objective is to enable students to make use of CFD in their cooperative-education work, senior capstone project as well as future engineering career. CFD has become an indispensable tool for engineering design & analysis, and it is now available to broad range of users, through commercial software packages. Proper use of these softwares, however, requires basic knowledge of CFD to understand their capabilities and limitations, to be aware of the pitfalls and to interpret the predictions. The course is designed to offer a balanced coverage of essential and applied CFD, with particular emphasis on verification & validation and CFD analysis. Training for a commercial CFD package is an integral part of the course which is facilitated by the use of project-based learning. In this presentation, details of development and implementation of this course will be discussed.

1:31PM R35.00003 ABSTRACT WITHDRAWN –

1:44PM R35.00004 Enhancing Student International Awareness and Global Competency through Compact International Experience Courses , FRANK JACOBITZ, THOMAS SCHUBERT, University of San Diego — Short-term, study-abroad, elective engineering courses were developed in order to raise the international awareness and global competency of engineering students. These Compact International Experience (CIE) courses were taught in response to a strong student desire for engineering study abroad courses and an effort by the home institution to internationalize its curriculum. An assessment of repeat offerings of two three-semester-unit courses on Topics in Fluid Mechanics and Advanced Electronic Circuit Design in a three-week time frame in France and Australia was performed. The goals of the two CIE courses are an effective teaching of their respective technical content as well as a student understanding of the cultural environment and the impact of engineering solutions from a global and societal viewpoint. In the repeat offerings, increased interaction with local industry was an additional goal. The CIE courses were assessed through surveys completed at the beginning and end of the courses, weekly student reflection papers, course evaluations, and formalized instructor observations. Based on the assessment performed, the two CIE courses have been found to be a valuable approach in the delivery of engineering technical electives combined with an international experience.

1:57PM R35.00005 Undergraduate ROV Outreach , KIP HACKING, RANDY HURD, GEOFF WRIGHT, TADD TRUSCOTT, Brigham Young University, SPLASH LAB TEAM — Grumpy smelly, and apathy stricken middle school students often find science to be "uncool" and "hands-off." We are changing this in our local area through an outreach program at ten participating middle schools building underwater remotely operated vehicles (ROV). Participants (300) were mentored by undergraduates (70) from BYU and instructed on various STEM topics including: electrical circuits, buoyancy, material science, design, and simple robotics. Through weekly visits these undergraduates provided teachers with immediate personal support to start their local ROV program, and enhanced their engineering curriculum at the middle school level. Each undergraduate also designed and built an instrument in an on-campus instrumentation class that were compatible with the younger student's ROVs. Designs, videos and building instructions were posted online for current and future student access. This project culminated in a timed competition where students from each school used their ROVs to collect dive rings and maneuver through an underwater obstacle course. In this talk we will discuss how to increase your own outreach efforts by connecting undergraduates with local K-12 students using inexpensive ROVs and instrumentation projects.

2:10PM R35.00006 Ocean Circulation in a Rotating Tank - An Outreach Project in Fluid Dynamics , SHANON RECKINGER, Fairfield University — A rotating water tank was designed and built by two senior mechanical engineering undergraduates at Fairfield University. The project was part of a year long senior design course. The rotating water tank is used to simulate oceanic and atmospheric phenomenon for classroom or outreach use. The following year, the tank was used for outreach as part of Fairfield University's Broadening Access to Science Education (BASE) camp. BASE camp is a two week residential camp for high school woman interested in scientific research. It is designed to inform and excite students by giving them a hands-on, research-based experience in the sciences, engineering, and mathematics. An all female research team composed of one mechanical engineering faculty member, two engineering undergraduates, and three high school students used the tank to explore "how the ocean moves." This talk will explain the design project and the outreach project in detail, in hopes of inspiring new fluids education and outreach ideas.

2:23PM R35.00007 Applying the results of education research to help students learn more: peer instruction and clicker questions in upper-division courses , RACHEL E. PEPPER, University of California Berkeley, STEPHANIE CHASTEEN, STEVEN POLLOCK, KATHERINE PERKINS, University of Colorado Boulder — Over the past 7 years, the physics faculty at the University of Colorado have worked to transform four upper-division courses: Classical Mechanics/Math Methods, Electricity and Magnetism (E&M) I and II, and Quantum Mechanics. We discuss our transformations as a model for other upper-division courses, such as fluid mechanics, focusing on one of the changes made in the transformation effort: the addition of peer instruction ("clicker questions") to lecture. The goals of our course transformation were to improve student learning and to develop materials and approaches that other faculty could easily adopt or adapt. In this talk, we review the evidence for effectiveness of peer instruction, discuss our implementation, and present evidence of improved student learning in our transformed upper division courses. Tips for effective use of peer instruction and banks of clicker questions available for fluid mechanics will also be discussed. Our curriculum materials are free and available at www.colorado.edu/sei/departments/physics.htm.

Tuesday, November 26, 2013 1:05PM - 3:15PM –

Session R36 Geophysical: Atmospheric IV 407 - Balaji Jayaraman, Pennsylvania State University

1:05PM R36.00001 Large-eddy simulations of the Ekman boundary layer over walls with variable temperatures: the complex implications of spatially-varying static stability[1] , ELIE BOU-ZEID, STIMIT SHAH, Princeton University, Department of Civil and Environmental Engineering — Understanding and parameterizing turbulent fluxes in statically-stable Ekman boundary layers (SABLs), where buoyant forces destroy turbulent kinetic energy, remains a challenging yet very important problem in geophysical fluid dynamics. The flow is further complicated when surface temperatures exhibit spatial variability leading to variation in buoyancy forces and to significant advection effects. A pertinent example is flow over polar leads and polynyas. To investigate the complex interactions of static stability and surface variability, large eddy simulations are performed over rough surface patches of different temperatures. Stable to more-stable, as well unstable-to-stable and stable-to-unstable transitions are simulated. Variability of surface temperature is shown to result in unexpected flow patterns: TKE is potentially higher under the more stable patches due to advection, and the subsidence and lofting of air over the different patches can counteract the effect of spatial TKE variability on the vertical fluxes. The TKE and flux budget development downstream of the surface temperature transition are investigates with the aim of developing models for upscaling the effect of surface heterogeneity under such conditions.

[1]This work is supported by NSF Physical and Dynamic Meteorology Program under AGS-1026636

1:18PM R36.00002 An LES Study of Transition in Atmospheric Boundary Layer Turbulence Structure from Neutral to Convective Stability States , BALAJI JAYARAMAN, JAMES BRASSEUR, Department of Mechanical Engineering, The Pennsylvania State University — The scales, strengths and detailed structure of atmospheric boundary layer (ABL) turbulence that affect wind turbine performance and reliability are strongly dependent on the relative contributions of buoyancy-driven vertical motions from surface heating and shear driven motions from geostrophic winds at the mesoscale, as characterized by the global stability state parameter $-z_i/L$. In the shear-dominant neutral limit, the ABL is characterized by streamwise-elongated coherent eddies of negative fluctuating horizontal velocity As surface heat flux is increased, buoyancy drives vertical fluctuations strongly correlated with shear-driven motions that eventually organize to generate streamwise rolls that couple upper with lower boundary layer. We use large-eddy simulation (LES) to study this transition between "near neutral" and "moderately convective" by quantifying correlations and integral scales as a function of -z_i/L.The interactions between outer and the surface layer eddies generate surprising turbulence dynamics that includes a special transitional stability state with unusually enhanced streamwise coherence. The transitional process includes a critical phenomenon with sudden dramatic change in ABL structure, and high sensitivity in horizontal fluctuations to surface heating at a low $-z_i/L$. *Supported by DOE.*

1:31PM R36.00003 Buoyancy effects on the mean velocity profile in atmospheric surface layer flows , SCOTT SALESKY, The Pennsylvania State University, GABRIEL KATUL, Duke University, MARCELO CHAMECKI, The Pennsylvania State University — Within the diabatic atmospheric surface layer (ASL), the mean velocity profile deviates from its conventional logarithmic shape by a stability correction function $\phi_m(\zeta)$ that varies with the stability parameter ζ. The ζ parameter measures the relative importance of mechanical to buoyant production or destruction of turbulent kinetic energy within the ASL. A theoretical framework is developed to link the coefficients in empirical curves for $\phi_m(\zeta)$ to stability-dependent properties of turbulence in the ASL including the variation with ζ of the integral lengthscales and anisotropy of momentum transporting eddies. Approximate asymptotic solutions for $\phi_m(\zeta)$ are derived for the slightly unstable and free convective limits in order to make an explicit link between $\phi_m(\zeta)$ and the stability variation of the integral lengthscale and eddy anisotropy. Analysis of data from the Advection Horizontal Array Turbulence Study indicates the theory is able to explain the behavior of $\phi_m(\zeta)$ accurately for slightly unstable and free convective conditions, but an explanation for the observed behavior of ϕ_m for stable conditions remains elusive.

1:44PM R36.00004 Underactuated (bouyancy) control of sensor vehicle distributions in highly stratified flows , GIANLUCA MENEGHELLO, THOMAS BEWLEY, Flow Control Lab, UC San Diego — Balloons and drifters are useful tools in observation and monitoring of the atmosphere and the ocean. In their simpler configuration, data acquisition is performed while they are passively transported by the flow and no control on their distribution is possible. We present a control strategy employing vertical (buoyancy) actuation only to control both their vertical and horizontal distribution, with application to hurricane monitoring. The desired horizontal distribution is obtained leveraging knowledge of the stratified flow velocity field. The optimal control framework is employed to compute the buoyancy time sequence driving the vehicles to the desired spatial distribution. Uncertainties in both the flow field description and the vehicles position are accounted for.

1:57PM R36.00005 Vertical concentration profiles of dust particles in the atmospheric surface layer[1] , LIVIA SOUZA FREIRE GRION, MARCELO CHAMECKI, Pennsylvania State University — The study of the emission of dust particles from soil surfaces into the atmospheric boundary layer (ABL) has important applications to different environmental problems, from local air quality to large-scale aerosol transport and its interaction with climate. Due to the difficulty of measuring surface dust flux, a model relating it to the vertical profile of mean concentration is needed. In this study, we use Large-Eddy Simulation of the ABL to evaluate the effects of particle size and turbulence on the relationship between dust flux and concentration profiles. Results show that for very small particles (less than 5 micrometers) the settling velocity is usually negligible and the mean concentration displays a logarithmic profile. For large particles (more than 30 micrometers), there is an approximate balance between vertical turbulent diffusion and gravitational settling, so that Prandtl's power-law solution holds. However, a more general solution including non-zero net fluxes and gravitational settling exists, and it is in agreement with LES results for all particle sizes. Effects of atmospheric stability are also investigated.

[1]Funding from the Science Without Borders program (CNPq, Brazil) is gratefully acknowledged.

2:10PM R36.00006 ABSTRACT WITHDRAWN –

2:23PM R36.00007 Modeling the structure and dynamics of a sea spray-stratified hurricane boundary layer[1] , YEVGENII RASTIGEJEV, North Carolina A&T State University, SERGEY A. SUSLOV, Swinburne University of Technology, Australia — Accurate modeling of the interaction between sea spray and a turbulent hurricane boundary layer is important for correct forecasting of the hurricane path and intensity. Here we applied a variable density (non-Bousinessq) E-epsilon turbulence closure model to describe the sea spray-stratified hurricane boundary layer structure and dynamics. The model accounts for the effects of variation of turbulent energy and turbulent mixing length due to the sea spray presence, and the spray inertia. The obtained results confirm that the influence of such variations is significant over the complete range of possible spray concentration values. It is shown that when the spray concentration is large the inertia effect dominates the momentum exchange close to the sea surface decelerating the airflow relative to the reference logarithmic profile. However, at higher altitudes the flow acceleration caused by the turbulence suppression is always observed provided that the spray droplets are sufficiently small. It was also found that the variable density model predicts a faster decrease of the drag coefficient with the flow speed than the Boussinesq model.

[1]NSF, award HRD-1036563

2:36PM R36.00008 On the Periodicity of Atmospheric von Kármán Vortex Streets[1] , CHRISTOPHER NUNALEE, SUKANTA BASU, North Carolina State University — For over one hundred years, a similarity relationship between Strouhal number (Sr), a non-dimensional metric for vortex shedding frequency (N), and Reynolds number (Re) has been aggressively pursued in the context of von Kármán vortex streets (VKVSs). In this study, we document the $Sr - Re$ relationship of atmospheric VKVSs (i.e., in the extremely high Re regime) in order to gain new insight into a regime of the similarity theory which has never before been investigated. Through quasi-idealized numerical simulations of realistic atmospheric VKVS events, we observe a range of Re in which mesoscale VKVSs are clearly present yet Sr remains in a steady range of 0.15 - 0.22 (irrespective of Re). This relationship resembles what has been observed for VKVSs in the much lower $10^2 < Re < 10^4$ regime suggesting eddy viscosity as a proxy for molecular viscosity with regards to $Sr - Re$ similarity theory at high-Re. In addition, we find the dominant length scale dictating the $Sr - Re$ relationship in the atmosphere to be the cross-stream mountain diameter, specifically at the height of the boundary layer thermal inversion.

[1]The authors acknowledge financial support received from the Department of Defense AFSOR grant under award number (FA9550-12-1-0449).

2:49PM R36.00009 Vortex roll-up in a stratified fluid , SURUPA SHAW, JOHN MCHUGH, None — Recent simulations of a vortex pair in a stratified fluid show that for small Froude number W/Nb the vortices disintegrate into internal waves, where W is the vortex strength, b is the vortex spacing, and N is the buoyancy frequency. The kinetic energy loss from the vortex pair in this regime can be remarkably fast, essentially annihilating the coherent vortex pair before any noticeable propagation. If the Froude number is large the vortices remain coherent and propagate as they would in constant density flow. The transition in behavior occurs near a Froude number of unity, but is apparently not a sharp transition, as some wave-making appears to happen for Froude numbers above unity. Here we quantify the wave-making with an integral of the momentum flux around a sequence of circles centered on the vortex pair and moving with it. Numerical solutions are obtained using a spectral method, the flow is treated as Boussinesq and viscous, and the initial conditions are approximately the flow due to a line vortex. The results confirm that the transition is gradual, although the complexity of the wavy flow makes interpretation difficult. These results are related to vortex roll-up in a stratified fluid.

3:02PM R36.00010 Finite-Amplitude Anelastic Internal Wave Transmission and Reflection in Non-uniform Flow and Stratification , LAUREN EBERLY, BRUCE SUTHERLAND, University of Alberta — Linear theory predicts that vertically propagating internal waves in non-uniform background flow and stratification reflect where their Doppler-shifted frequency matches the buoyancy frequency at a particular height. If the height over which the waves are evanescent is small, the waves can tunnel, partially transmitting their energy above the reflection level. Furthermore, if the waves grow sufficiently in amplitude due to anelastic effects, weakly nonlinear effects (specifically, the transient acceleration of the mean winds by the wave-induced mean flow) can further enhance transmission. These dynamics are examined through fully nonlinear simulations of anelastic waves in retrograde shear and in non-uniform stratification. For non-hydrostatic waves that are modulationally unstable, we find that transmission is enhanced across a reflection level provided it is situated sufficiently high that weakly nonlinear effects become important but not so high that the waves overturn before reaching the reflection level. More hydrostatic, modulationally stable, waves have enhanced dispersion and so behave more like linear theory predictions except that their overturning heights can be many density scale heights above the predicted level.