65th Annual Meeting of the APS Division of Fluid Dynamics
San Diego, California
http://www.aps.org/meetings/meeting.cfm?name=DFD12

Sunday, November 18, 2012 8:00AM - 9:44AM –

Session A1 Fluid Structure Interactions I 22 - Chair: Kourosh Shoele, RE Vision Consulting

8:00AM A1.00001 Limits of the potential flow model for obstacle detection using a lateral line AUDREY MAERTENS, MIT, GABRIEL WEYMOUTH, SMART, MICHAEL TRIANTAFYLLOU, MIT — Fish have a particular sensory system called lateral line through which they measure flow velocity and pressure gradient. Behavioral studies have shown that fish can detect and identify obstacles while gliding using this sensory system alone. Despite a widespread interest of the community in understanding and reproducing this capability, a realistic approach is still missing. Indeed, due to computational constraints, most attempts to date have used potential flow models. The present work aims at revealing the limits of the potential flow model in the case of a vehicle gliding by a cylinder. The understanding thus gained can be used to account for viscous effects in a computationally-efficient fashion. An improvement of the boundary data immersion method provides accurate pressure predictions at the Reynolds numbers considered ($500 < \mathrm{Re} < 10000$). It is shown how a potential flow-based obstacle detection algorithm fails at locating the cylinder at these Reynolds numbers. It is also shown that a panel method accounting for dynamically changing displacement thickness leads to accurate pressure prediction. This is a first step toward real-time pressure predictions for viscous flows which is needed for efficient obstacle detection and identification algorithms.

8:13AM A1.00002 A finite volume algorithm for fluid-structure interaction problems using unstructured meshes , BRANISLAV BASARA, AVL List GmbH — The coupled simulation between multiple fluid and solid domains plays an important role in a wide range of multi-physics problems. A simple method is to calculate different computational domains separately by submitting one or more executable codes for each calculation domain and even for solving different physics, and then exchanging data with the so called coupling server which could use a direct data exchange or in-directly via data files. However, nowadays, sophisticated numerical techniques allow separate domains to be solved with the same computational code within one calculation run. The finite volume method based on the collocated variable arrangement and adopted for the general polyhedral control volume is extended here to solve beside fluid flow problems also deformations and stresses in the solid structure. The performance of the method will be demonstrated on a number of fluid flow and stress analysis test cases. Results show that the method can be used as a useful tool for solving fluid-structure interaction problems.

8:26AM A1.00003 A numerical framework for modelling floating wind turbines[1] , AXELLE VIRE, JIANSHENG XIANG, MATTHEW PIGGOTT, JOHN-PAUL LATHAM, CHRISTOPHER PAIN, Imperial College London — This work couples a fluid/ocean- and a solid- dynamics model in order to numerically study fluid-structure interactions. The fully non-linear Navier-Stokes and solid-dynamics equations are solved on two distinct finite-element and unstructured grids. The interplay between fluid and solid is represented through a penalty force in the momentum balances of each material. The present algorithm is novel in that it spatially conserves the discrete penalty force, when exchanging it between both models, independently of the mesh resolution and of the shape-function orders in each model. This numerical framework targets the modelling of offshore floating wind turbines. Results will be shown for the flow past a moving pile and an actuator-disk representation of a turbine.

[1]This research is supported by the European Union Seventh Framework Programme (grant agreement PIEF-GA-2010-272437)

8:39AM A1.00004 Dynamic evolution of a flow to localized, kinetics-driven ablation or coagulation[1] , DANIEL HAGAN, RYAN CROCKER, YVES DUBIEF, University of Vermont — This research focuses on the numerical simulation of the ablative creation of a cavity or a coagulative formation at a wall in a flow. The fluid-solid interface is defined by a level set (LS) variable, whose transport equation is driven by the mass-loss or growth process. The boundary conditions at the fluid-solid interface are enforced by a mass and energy-conserving immersed boundary method (IBM) using the ghost-fluid node approach for the latter and for the transport of chemical species. The first application of the LS/IBM algorithm is a channel flow in which both walls are cavity-free, but one wall contains a section made of ablatable material, which could correspond to a hole or gap in a spacecraft thermal protection shield. The second application is a pipe flow in which the wall is capable of accumulating material, which could describe the coagulation of blood at a vessel wall. The solid mass loss or growth is driven by one step kinetics. For both flows, the dynamical interplay between the ablative or coagulative patch is investigated through statistics and flow topology.

[1]We gratefully acknowledge the financial support of NASA, grant No. NNX11AM07A, and NIH, grant No. P01HL46703, and the computational support of the Vermont Advanced Computing Core.

8:52AM A1.00005 High Order Solution of the Incompressible Navier-Stokes Equations in Immersed Domains[1] , JEAN-CHRISTOPHE NAVE, McGill University, ALEXANDRE MARQUES, RUBEN ROSALES, MIT — The Correction Function Method (CFM) is a general framework that can be used to devise highly accurate discretizations for diffusion dominated phenomena in the presence of immersed interfaces or boundaries. In previous work, the authors presented the CFM for the solution of the Poisson equation. In this talk, we discuss the application of the CFM to time-dependent problems, with emphasis on the incompressible Navier-Stokes equations. Fourth-order accurate results are presented.

[1]The authors acknowledge support from the NSF and NSERC.

9:05AM A1.00006 Numerical Capture of Wing-tip Vortex Using Vorticity Confinement , BAILI ZHANG, JING LOU, CHANG WEI KANG, Instiute of High Performance Computing, Singapore, ALEXANDER WILSON, Rolls-Royce Marine, UK, JOHAN LUNDBERG, Rolls-Royce Hydrodynamic Research Centre, Sweden, RICKARD BENSOW, Chalmers University of Technology, Sweden — Tracking vortices accurately over large distances is very important in many areas of engineering, for instance flow over rotating helicopter blades, ship propeller blades and aircraft wings. However, due to the inherent numerical dissipation in the advection step of flow simulation, current Euler and RANS field solvers tend to damp these vortices too fast. One possible solution to reduce the unphysical decay of these vortices is the application of vorticity confinement methods. In this study, a vorticity confinement term is added to the momentum conservation equations which is a function of the local element size, the vorticity and the gradient of the absolute value of vorticity. The approach has been evaluated by a systematic numerical study on the tip vortex trailing from a rectangular NACA0012 half-wing. The simulated structure and development of the wing-tip vortex agree well with experiments both qualitatively and quantitatively without any adverse effects on the global flow field. It is shown that vorticity confinement can negate the effect of numerical dissipation, leading to a more or less constant vortex strength. This is an approximate method in that genuine viscous diffusion of the vortex is not modeled, but it can be appropriate for vortex dominant flows over short to medium length scales where viscous diffusion can be neglected.

9:18AM A1.00007 Implementation of a Phase-Lagged Boundary Condition for Turbomachinery[1] , ALEX WOUDEN, JOHN CIMBALA, BRYAN LEWIS, Pennsylvania State University — One factor that contributes significantly to the cost of a time-dependent CFD simulation is the size and scope of the computational domain. Common approximations, such as periodic and symmetric boundary conditions, have the advantage of reducing the domain proportional to its periodicity or symmetry. However, turbomachinery applications featuring multiple blade rows render the periodic boundary condition unphysical because the adjacent blade rows are designed with dissimilar blade counts. Though the meshes of adjacent blade rows can be modeled independently and data can be interpolated across a grid interface, applying the standard periodic definition to the coupled faces leads to an over-constrained situation: a failure to reconcile two governing relations imposed on the same cell. A phase-lagged boundary condition (PLBC) relaxes the over-constraint problem and provides a more correct assumption for the resulting flow field. PLBC is available in a limited number of private CFD codes and is only briefly documented in the literature. The present work expands upon its development by implementing PLBC in OpenFOAM®, an open-source CFD software package. Its performance is demonstrated for basic turbomachinery applications through comparisons with full-wheel simulation.

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

9:31AM A1.00008 Characterization of the Boundary Conditions at the Test Section Inlet for a Combustion Rig , CHRISTOPHER RUSCHER, JOHN DANNENHOFFER, III, MARK GLAUSER, Syracuse University, BALU SEKAR, VINCENT BELOVICH, Air Force Research Lab — Large eddy simulations are sensitive to boundary conditions and therefore it is important to characterize the boundary conditions. The boundary conditions for the test section inlet of a combustion rig have been calculated using a computational fluid dynamic (CFD) simulation. A simulation was performed in lieu of experimental testing due to the complexity and cost of placing sensors in the upstream portions of the rig. The mean and RMS profiles for velocity as well as auto-spectrum are computed for the inlet. The calculated values will be used in future simulation work for this combustion rig.

Sunday, November 18, 2012 8:00AM - 9:44AM –

Session A2 Convection and Buoyancy-Driven Flows I: Rayleigh-Bénard Convection 23A - Chair: Eric Brown, University of California, Merced

8:00AM A2.00001 Comparison between 2D and 3D turbulent Rayleigh-Benard convection , ERWIN VAN DER POEL, RICHARD STEVENS, DETLEF LOHSE, University of Twente — We provide a detailed comparison between the results for 3D and the computationally much cheaper 2D numerical simulation of Rayleigh-Bénard (RB) flow. The 2D character of the LSC in both 2D and 3D flows hints towards a large similarity, in particular for integral quantities. Moreover, theoretical concepts for the scaling relations in turbulent RB convection are not specific to 2D or 3D. We compare the global transport quantities, boundary layer profiles, and flow topology of 2D and 3D convection to find at which control parameters differences and similarities can be found. One of the pronounced differences is the limited motion of the LSC in 2D, which increases the accumulation of energy in corner-rolls, leading to large scale wind reversals and high sensitivity of global output parameters on the roll-state.

8:13AM A2.00002 Supercritical Quasi-conduction States and Multiple Stable Flows in Stochastic Rayleigh-Bénard Convection , DANIELE VENTURI, GEORGE KARNIADAKIS, Brown University — Stochastic bifurcations and stability of natural convection within two-dimensional square enclosures are investigated by using different stochastic modeling approaches. Deterministic stability analysis is carried out first to obtain steady state solutions and primary bifurcations. It is found that multiple stable steady states coexist, in agreement with recent results, within specific ranges of Rayleigh number. Stochastic simulations are then conducted around bifurcation points and transitional regimes. The influence of random initial conditions and random perturbations in the temperature distribution at the horizontal walls is also investigated. It is found that random noise renders the bifurcation process to convection imperfect and it extends the range of stability of quasi-conduction states beyond the classical onset of convection. In particular, subcritical and nearly supercritical quasi-conduction stable states are observed within the range of Rayleigh numbers Ra=0-4000. Analysis of the stochastic bifurcation diagrams also shows the presence of a stochastic drift phenomenon in the heat transfer coefficient, especially in the transcritical region. Such stochastic drift is investigated further by means of a sensitivity analysis based on functional ANOVA decomposion.

8:26AM A2.00003 Geometry-dependence of dynamics of the large-scale circulation in turbulent Rayleigh-Benard convection , ERIC BROWN, RUSSELL HAWKINS, University of California, Merced — We present a simple model for the geometry-dependence of dynamics of the large-scale circulation (LSC) in turbulent Rayleigh-Benard convection. This extends a previous model of stochastic ordinary differential equations (ODEs) developed to describe LSC dynamics in an aspect ratio 1 upright cylinder. The large-scale circulation exhibits behavior such as spontaneous meandering of the orientation, cessations of the flow, and various oscillation modes, which can change with the geometry of the container. Specifically, in experiments with a sideways cylinder the LSC orientation is found to align with the longer diameters, and new oscillation modes around and between these longest diagonals were found. These changes in dynamics with geometry can be accounted for with the addition of a shape-dependent potential term to the stochastic ODE model that represents the pressure forcing from the sidewall. For the sideways cylinder, the potential is analogous to a Duffing oscillator. The agreement between model predictions and observations suggests the possibility of describing LSC flows in arbitrary geometries with a relatively simple stochastic ODE model.

8:39AM A2.00004 Comparison between rough and smooth plates within the same Rayleigh-Benard cell , ELEONORE RUSAOUEN, JULIEN SALORT, FANNY SEYCHELLES, JEAN-CHRISTOPHE TISSERAND, ENS Lyon, MATTHIEU CREYSSELS, LMFA, OLIVIER LIOT, BERNARD CASTAING, FRANCESCA CHILLA, ENS Lyon — A Rayleigh-Benard cell consist in a tank filled of a fluid on which a temperature difference is imposed thanks to a cold plate at top and a hot at bottom. Movement is induced by the buoyancy force. Considering most of experimental apparatus previously used all around the world, both plates are smooth. Recently, the effect of roughness on thermal transfer had become a subject of interest. The present experiment is an asymetrical rough Rayleigh-Benard cell. Indeed the hot plate is rough whereas the cold plate is still smooth. Previously, tests conducted with 2mm high roughness showed independence of the two plates and a heat flux enhancement on the rough plate, which appeared to be greater than expected from the surface increase. This regime was caracterized by a $Nu \propto Ra^{1/2}$ law. New results obtained with a 4mm high roughness also show this flux enhancement and the independent behaviour of the plates. But a transition appears at high Rayleigh from the 1/2 power law regime to a 1/3 one. Former results obtained in the same symetrical smooth/smooth cell also showed a 1/3 law. But the rough 1/3 regime reveals a multiplier coefficient of 1.6 with the smooth one.

8:52AM A2.00005 Lagrangian tracking of an instrumented particule in Rayleigh-Benard flow , FANNY SEYCHELLES, ENS Lyon, XAVIER RIEDINGER, EXETER, JULIEN SALORT, ELEONORE RUSAOUEN, ENS Lyon, MATTHIEU GIBERT, LEGI, YOANN GASTEUIL, OLIVIER LIOT, BERNARD CASTAING, FRANCESCA CHILLA, ENS Lyon — Thermal convection is present in different systems from astrophysical to geophysical flows. Most experiments and numerical studies are carried from eulerian point of view. The heat transfer from a local perspective is not well understood. Lagrangian description could provide the missing insights on the local mixing and transport mechanisms in thermal convection. We present here a lagrangian measurement in a Rayleigh Bénard convection. To be more precise, we have conceived a sphere particle with embarked thermometers and radio emitter. Our experimental setup is a rectangular vessel with height H=40 cm and section 40 cm x 10 cm filled with water. The walls are made of polymethylmetacrylate, the top plate is cooled by controlled thermal bath, and the bottom plate is heated by electrical resistance. Using a camera for optical tracking, we obtain at the same time the position and temperature measurement of the particle. We present here the results of a new sensor on statistics of temperature, velocity and heat transport in an original Rayleigh Benard convection, where the top plate is smooth and the bottom plate is rough.

9:05AM A2.00006 Spatial Dependence of Boundary Layers in Simulations of Rayleigh-Bénard Convection , JANET SCHEEL, JORGE MUNOZ, Occidental College — We present results from a systematic study of the thermal and viscous boundary layers obtained from simulations of Rayleigh-Benard convection in a cylindrical container of aspect ratio one (diameter = depth). We extract local boundary layers at a variety of locations in the container for Rayleigh numbers between $1 \times 10^5 - 1 \times 10^9$ and for Prandtl numbers from $0.021 - 0.7$. We find some spatial dependence of boundary layer thicknesses, but these results highly depend on the method of extracting boundary layer thickness. We find similar results for the scaling of boundary layer thickness with Rayleigh number. This is an extension of our study of boundary layers performed in the center of the cell (J.D. Scheel et al., JFM, 2012). We compare our results to others, including the work by Wagner, et al., JFM, 2012.

9:18AM A2.00007 Reynolds number measurements for turbulent Rayleigh-Bénard convection with $0.17 < Pr < 0.88$[1] , JAMES HOGG, GUENTER AHLERS, UC Santa Barbara — We report Reynolds-number measurements from space-time cross-correlation functions of shadowgraph images taken of turbulent Rayleigh-Bénard convection in a cylindrical cell of height $L = 9.5$ mm and aspect ratio $\Gamma = 10.6$. The fluids were pure gases with Prandtl-numbers $Pr \approx 0.7$ and gas mixtures with $0.17 \leq Pr < 0.7$. The Rayleigh-number range was $10^5 \leq Ra \leq 10^8$. The elliptic approximation of He and Zhang[2] was used to calculate the mean flow velocity U and the rms fluctuation velocity V. For this system U was close to zero, and the Reynolds number Re_V based on V had Ra- and Pr-dependences consistent with the Grossmann-Lohse model.[3]

[1]Work supported by NSF grant DMR11-58514.

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

[3]S. Grossmann and D. Lohse, Phys. Rev. E, **66**, 016305 (2002).

9:31AM A2.00008 Heat transport by turbulent Rayleigh-Bénard convection for $Pr \simeq 0.8$ and $4 \times 10^{11} < Ra < 2 \times 10^{14}$: ultimate-state transition for aspect ratio $\Gamma = 1.00$ [1] , DENNIS P.M. VAN GILS, XIAOZHOU HE, MPI Dynamics and Self-Organization, Goettingen, Germany, DENIS FUNFSCHILLING, LRGP CNRS, Nancy, France, GUENTER AHLERS, UCSB, Santa Barbara, USA, EBERHARD BODENSCHATZ, MPI Dynamics and Self-Organization, Goettingen, Germany — We report experimental results for heat transport by Rayleigh-Bénard convection (RBC) in a cylindrical sample with aspect ratio $\Gamma \equiv D/L = 1.00$ ($D = 1.12$ m is the diameter and $L = 1.12$ m is the height) over the range $4 \times 10^{11} < Ra < 2 \times 10^{14}$ at $Pr \simeq 0.8$. For $Ra < Ra_1^* \simeq 2 \times 10^{13}$ we find $Nu = N_0 Ra^{\gamma_{\rm eff}}$ with $\gamma_{\rm eff} = 0.321 \pm 0.002$ and $N_0 = 0.0776$, consistent with classical turbulent RBC in a system with laminar boundary layers (BLs) below the top and above the bottom plate and with the prediction of Grossmann and Lohse[(1)]. For $Ra > Ra_1^*$ the data rise above the classical-state power-law and show greater scatter. In analogy to similar behavior observed for $\Gamma = 0.50$[(2)], we interpret this phenomenon as the onset of the transition to the ultimate state. Within our resolution this onset occurs at nearly the same value of Ra_1^* as it does for $\Gamma = 0.50$.

[1]Supported by the Max Planck Society, the Volkswagen Stiftung, the DFD Sonderforschungsbereich SFB963, and NSF grant DMR11-58514.

Sunday, November 18, 2012 8:00AM - 9:31AM –

Session A3 Multiphase Flows: Cavitation 23B - Chair: Martin Wosnik, University of New Hampshire

8:00AM A3.00001 Velocimetry in both phases of a cavitating flow by fast X-ray imaging[1] , OLIVIER COUTIER-DELGOSHA, ILYASS KHLIFA, LML / Arts et Metiers ParisTech, MARKO HOCEVAR, Laboratory for water and turbine machines, Faculty of Mechanical Engineering, Univ. of Ljubljana, SYLVIE FUZIER, LML / Arts et Metiers ParisTech, ALEXANDRE VABRE, CEA LIST, KAMEL FEZZAA, X-ray Science Division, Argonne National Laboratory — A promising method to measure velocity fields in a cavitating flow is presented. Dynamics of the liquid phase and of the bubbles are both investigated. The measurements are based on ultra fast X-ray imaging performed at the APS (Advanced Photon Source) of the Argonne National Laboratory. The experimental device consists of a millimetric Venturi test section associated with a transportable hydraulic loop. Various configurations of velocity, pressure, and temperature have been investigated. Radio-opaque particles are used as tracers for the liquid phase, in association with a multi-pixels sensor to record the successive positions of the particles. The use of X-rays instead of light solves the problem of light reflection and dispersion on phase boundaries, since X-rays penetrate a gas/liquid flow in straight lines. Images contain simultaneously the information related to the particles (for PIV analysis in the liquid), to the vapor bubbles (for PIV in the gas). The slip velocity between vapor and liquid is calculated everywhere both velocities can be obtained.

[1]Use of the Advanced Photon Source at Argonne National Laboratory was supported by the U. S. Department of Energy, Office of Science, and Office of Basic Energy Sciences.

8:13AM A3.00002 Time Resolved 2D X-ray Densitometry of a Cavitating Wedge[1] , SIMO MAKIHARJU, HARISH GANESH, STEVEN CECCIO, University of Michigan — A canonical cavitating two dimensional wedge experiment was setup to provide quantitative validation quality data for CFD through the use of time resolved x-ray densitometry. The x-ray system was used to the measure the spatial distribution of void fraction of the cavitating flow. The water tunnel's dissolved gas content was controlled and dissolved oxygen content measured. The flow was nominally two dimensional, and time resolved 2D projections of the void fraction distribution were measured with resolution of the order of $1/100^{th}$ of the cavity length. FFT of void fraction revealed the frequencies inherent to the cavitating wedge at Strouhal number on the order of 0.3. The pressure in the region of cavitation was recorded simultaneously with the x-ray measurement to compare the detected frequencies. As ongoing work, we are incorporating a 2D electrode array onto the surface of the model to have a second measure of the void fraction within ~1 mm of the surface, thus providing a consistency check for the x-ray densitometry and revealing any near surface 3D features of this nominally 2D flow. The data covered the incipient sheet cavity, intermittent cloud shedding and fully shedding periodic cavity regions for a range of cavitation numbers.

[1]Research supported by the Office of Naval Research

8:26AM A3.00003 Large eddy simulation of cavitating mixing layer: the effect of splitter plate , KAMESWARARAO ANUPINDI, DINESH SHETTY, STEVEN FRANKEL, Purdue University — The accurate prediction and validation of various cavitation mechanisms such as internal jets, cavity extent etc., play an important role in furthering the reliability of simulations. Previous simulation studies[1] point out a need for high resolution solver. Therefore, in the present study we perform high order large eddy simulation (LES) study on a turbulent incompressible mixing layer operating under cavitating and non-cavitating conditions. The solver is based on incompressible Navier-Stokes formulation, in which a single fluid, two-phase mixture is solved with a separate liquid volume fraction (α_l) equation coupled with a mass-transfer model. We first validate our solver by making comparisons to experimental results[2]. Various quantities such as similarity velocity profiles, mean vapor fraction ratio profiles, and the evolution of vorticity thickness are compared. Having validated the solver, we include the splitter plate in the simulations through immersed boundary method approach and study the effect of the wake created by it on the cavitating mixing layer dynamics.

[1]Bensow et al. J. Fluids Engg., **132**, 041302 (2010)
[2]Aeschlimann et al. Phys. Fluids, **23**, 055101 (2011)

8:39AM A3.00004 Towards DNS/LES of cavitating flows in complex geometries[1] , ASWIN GNANASKANDAN, KRISHNAN MAHESH, University of Minnesota — We are developing a numerical method for DNS/LES of turbulent cavitating flows in complex geometries. The multiphase medium is represented using a homogeneous equilibrium model that assumes thermal equilibrium between the liquid and the vapor phase. The governing equations are the compressible Navier Stokes equations for the liquid/vapor mixture along with a transport equation for the vapor mass fraction. A separate total energy equation is solved, as opposed to assuming isothermal flow. The unstructured compressible algorithm in (Park & Mahesh, AIAA Paper 2007-0722) has been extended to solve for multiphase flows. A characteristic filter based shock capturing scheme, extended to handle non-ideal gases and mixtures, is applied in a predictor-corrector approach, ensuring that the shock-capturing is active only in the regions of discontinuity. A segregated implicit method is used to address the stiffness of the system. We discuss our numerical method, validation using benchmark problems and its application to study cavitation behind a circular cylinder for three different cavitation numbers σ = 2.0, 1.0 and 0.7.

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

8:52AM A3.00005 Cavitation Characteristics of a NACA 63-424 Hydrofoil and Performance Comparison with a Bidirectional Version of the Foil , IVAYLO NEDYALKOV, MARTIN WOSNIK, University of New Hampshire — A NACA 63-424 hydrofoil with a 75mm chord and a 152mm span was tested in the recently renovated 6-inch high-speed water tunnel at the University of New Hampshire. The NACA 63-424 foil is being considered for use on rotors of marine hydrokinetic turbines, including the US Department of Energy Reference Horizontal Axis Turbine (RHAT) for tidal and ocean current applications. For various angles of attack, the foil was tested at speeds ranging from 2m/s to 12m/s. Pressure in the test section was varied independently. For each angle, speed and pressure setting, high speed videos were recorded (at 3600 frames per second and above). Cavitation inception and desinance were obtained. Lift and drag were measured using a new 2-component force balance. In tidal turbines applications, bidirectional foils do not require pitch control, hence the experiments were repeated for a bidirectional version of the NACA 63-424 foil and the characteristics of the two foils were compared. The results can be used to predict cavitation inception and performance of marine hydrokinetic turbines, for a given site, deployment depth and and tip speed ratio.

9:05AM A3.00006 Large Eddy Simulation of cavitation in turbulence , SERGEI CHUMAKOV, DAVID COOK, Robert Bosch LLC, FRANK HAM, Cascade Technologies, UWE IBEN, Robert Bosch GmbH — Large Eddy Simulation of a turbulent cavitating flow has been performed using the explicit spatially-filtered compressible Navier-Stokes solver. The unstructured finite volume method uses a blended central-upwind scheme in single-phase regions to minimize artificial damping of resolvable turbulence scales. In the areas with discontinuities such as phase change, the method switches to a lower-order reconstruction (WENO and first order) and an approximate Riemann solver. Time discretization is performed with an explicit third order Runge-Kutta scheme. Comparison of our results for several cases to simulations and experiments from the current literature is presented.

9:18AM A3.00007 Axisymmetric, Ventilated Supercavitation in Unsteady, Horizontal Flow , ELLISON KAWAKAMI, SEUNG-JAE LEE, ROGER ARNDT, Saint Anthony Falls Laboratory - University of Minnesota — 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, an artificial supercavity is required until the vehicle can reach conditions at which a natural supercavity can be sustained. Previous studies at Saint Anthony Falls Laboratory (SAFL) focused on the behavior of ventilated supercavities in steady horizontal flows. In open waters, vehicles can encounter unsteady flows, especially when traveling under waves. A study has been carried out at SAFL to investigate the effects of unsteady flow on axisymmetric supercavities. An attempt is made to duplicate sea states seen in open waters. In an effort to track cavity dimensions throughout a wave cycle, an automated cavity tracking script has been developed. Using a high speed camera and the proper software, it is possible to synchronize cavity dimensions with pressure measurements taken inside the cavity. Results regarding supercavity shape, ventilation demand, cavitation parameters and closure methods are presented. 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. The supercavity volume varied with cavitation number and a possible relationship between the two is being explored. (Supported by ONR)

Sunday, November 18, 2012 8:00AM - 9:44AM –

Session A4 Drops I 23C - Chair: Stephen Morris, University of California, Berkeley

8:00AM A4.00001 Effects of microscale inertia on heat or mass transfer from a drop , DEEPAK KRISHNAMURTHY, GANESH SUBRAMANIAN, Jawaharlal Nehru Center for Advanced Scientific Research, Bangalore — Heat or mass transport from suspensions of solid particles or drops is ubiquitous in many industrial processes. In the zero inertia limit the transport is diffusion limited owing to the presence of closed streamlines around each particle. A small but finite amount of inertia though, results in a vastly different picture, greatly enhancing transport by destroying the closed streamline configuration. We develop a theoretical formulation to study the effects of weak inertia on transport from a density-matched drop in a 2D linear flow. It is shown that, unlike a solid particle, the near-surface streamlines are closed only when the viscosity ratio (λ) exceeds a critical value $\lambda_c = 2\alpha/(1-\alpha)$, where α is the linear flow parameter measuring relative magnitudes of extension and vorticity. The velocity field on the drop surface can be characterized using a complex-valued analogue of the (C,τ) coordinate system used to describe Jeffrey orbits of an axisymmetric particle. In the open-streamline case ($\lambda < \lambda_c$), convective transport occurs even with zero inertia, and for large Peclet number (Pe) (the relative magnitude of convective to diffusive transport), the Nusselt number (dimensionless rate of heat transfer) is expected to scale as $F(\alpha,\lambda)Pe^{1/2}$ and is determined via a boundary layer analysis in the (C,τ) coordinate system. In the closed streamline case ($\lambda > \lambda_c$), similar to the solid particle, inertia plays a crucial role, and the Nusselt number must scale as $G(\alpha,\lambda Re^{1/2}Pe^{1/2}$. A methodology is developed to analyze the convection along spiraling streamlines using a physically motivated choice of coordinate system on the drop surface.

8:13AM A4.00002 When a water drop freezes before it solidifies , PIROUZ KAVEHPOUR, University of California, Los Angeles, STEPHEN DAVIS, Northwestern University, FARYAR TAVAKOLI, University of California, Los Angeles — When a drop of liquid is placed on a substrate which temperature is below the melting point of the liquid, one would expect the drop to solidify instantaneously. However, many liquids, such as water, must be subcooled to solidify below its melting temperature due to homogeneous nucleation's high activation energy. Most of the drop solidification research, particularly for water, phase change is assumed to occur at equilibrium freezing temperature; however, this is not the case. We found that after a certain degree of supercooling, a kinetic based nucleation begins and latent heat of fusion is suddenly liberated, causing an increase in liquid temperature. At the end of this stage, approximately 20% of the drop is crystallized. This phenomenon is known among metallurgists as recalescence. This is followed by a slow solidification process at the melting point. As a water droplet spreads on a cold substrate, its contact line stops just prior to freezing inception from the liquid-solid interface. In this study, we assert that recalescence prior to solidification may be the cause of water's sudden immobility, which results in a fixed contact angle and droplet diameter. In our experiments, the nucleation front initiates from the trijunction point and propagates to the drop volume.

8:26AM A4.00003 Fluid Dynamics of Condensed Droplets on Hybrid Surfaces , CHUN WEI YAO, JORGE ALVARADO, Texas A&M University, CHARLES MARSH, Engineer Research and Development Center, U.S. Army Corps of Engineers, ANTHONY JACOBI, University of Illinois at Urbana-Champaign, TEXAS A&M TEAM, ERDC/UIUC TEAM — Droplets impinging on hybrid surfaces consisting of a micropillar array of hydrophobic and hydrophilic sites exhibit a distinct wetting behavior that can be estimated using a surface energy-based model. However, condensed droplets display a quite different wetting behavior when they grow on hybrid surfaces. In this study, hybrid surfaces with four different spacing ratios were tested in a condensation cell. For hybrid surfaces with spacing ratio below 2, initial droplets form on the top of the micropillars, and then grow and coalesce with adjacent droplets, which shed after they reach a given size. After shedding, the top surface remains partially dry which allows for faster droplet re-nucleation. Droplet growth and coalescence continue until very large droplets are formed which is a characteristic behavior of Wenzel droplets For hybrid surfaces with spacing ratio equal to 2, a quite interesting wetting behavior is observed when droplets coalesce. The coalesced droplets form a thin liquid film, which sheds eventually under the right conditions. The results indicate that spacing ratio is the key factor for having different fluid dynamics of condensed droplets on hybrid surfaces.

8:39AM A4.00004 Evaporation control of a drop on fibers , CAMILLE DUPRAT, ALISON D. BICK, HOWARD A. STONE, Princeton University — The evaporation dynamics of mist from a fibrous material has important industrial and environmental consequences. In order to understand the drying kinetics of a fiber array, we study a drop sitting on two parallel fibers and, in particular, investigate how the structure and mechanical properties of the material affect the drying dynamics. A drop deposited on two parallel fibers can adopt different shapes, depending on the drop volume, the interfiber distance, the surface tension of the liquid and the fiber radius and elasticity. A given volume of liquid on a given pair of fibers can adopt two distinct morphologies. These two states exhibit different drying kinetics that greatly affects the overall drying time. In addition, flexible fibers undergo deformation that leads to spontaneous spreading, hence to an increased drying rate. The drying kinetics is thus controlled by the fiber elasticity. Furthermore, during evaporation, the drop can change morphology, which counterintuitively results in an increased wetted area during drying.

8:52AM A4.00005 Directional motion of evaporating droplets on gradient surfaces[1] , SHUHUAI YAO, LI XU, ZHIGANG LI, The Hong Kong University of Science and Technology — Droplet evaporation on surfaces has various applications in drying problems such as ink-jet printing, pesticide spraying, chemical or biological detection, and DNA microarray spotting technology. Controlling evaporating droplets via substrate morphology and/or wetting properties allows for efficient deposition of sample molecules in these applications. In this work, evaporation of sessile water droplets on surfaces with wettability gradients was studied. The wettability gradient was generated by fabricating non-uniformly distributed cylindrical micropillars on silicon surfaces. During the evaporation, it was found, along the wettability gradient, that the contact line on one side was strongly pinned, while the contact line on the other side depinned and gradually receded, making the center of mass of the droplet move either in or against the direction the wettability gradient, depending on the configuration of the micropillars. The theoretical criterion predicting the moving direction was derived based on the excess free energy and the energy barrier during the evaporation. The theoretical predications agreed well with the experimental observations. The results provide a parametric design basis to control the contact line dynamics and directional transport of evaporating droplets.

[1]This work was supported by the Research Grants Council of Hong Kong under General Research Fund (Grant No. 621110).

9:05AM A4.00006 Apparent contact angle of an evaporating drop , S.J.S. MORRIS, U.C. Berkeley — In experiments by Poulard et al. (2005), a sessile drop of perfectly wetting liquid evaporates from a non–heated substrate into an under–saturated mixture of vapour with an inert gas; evaporation is limited by vapour diffusion. The system exhibits an apparent contact angle θ that is a flow property. Under certain conditions, the apparent contact line was stationary relative to the substrate; we predict θ for this case. Observed values of θ are small, allowing lubrication analysis of the liquid film. The liquid and vapour flows are coupled through conditions holding at the phase interface; in particular, vapour partial pressure there is related to the local value of liquid pressure through the Kelvin condition. Because the droplet is shallow, the interfacial conditions can be transferred to the solid–liquid interface at $y = 0$. We show that the dimensionless partial pressure $p(x, y)$ and the film thickness $h(x)$ are determined by solving $\nabla^2 p = 0$ for $y > 0$ subject to a matching condition at infinity, and the conditions $-p = \mathcal{L}h_{xx} + h^{-3}$ and $(h^3 p_x)_x + 3p_y = 0$ at $y = 0$. The parameter $\mathcal{L}$ controls the ratio of Laplace to disjoining pressure. We analyse this b.v.p. for the experimentally–relevant case $\mathcal{L} \to 0$.

9:18AM A4.00007 Thermal Effects of the Substrate on Water Droplet Evaporation , BENJAMIN SOBAC, DAVID BRUTIN, IUSTI Lab, Aix-Marseille Université — Since a few decades, the evaporation of a drop deposited onto a substrate has been subject to numerous research activities due to the increase of the range of applications underpinned by this phenomenon. However, this process today is always a challenging problem in soft matter physics due to the complexity of present couplings: fluid dynamic, physical chemistry of the substrate, heat and mass transfer. The originality of the presented experiment is to decouple the effects of wetting properties and thermal properties of the substrate. Thus, whereas we previously presented the role of wetting properties on evaporation by changing the surface energy and the roughness while maintaining the thermal properties constant thanks to nanoscale coatings on the substrate surface (B. Sobac and D. Brutin, Langmuir 27, 14999 (2011)), we investigate here the influence of the thermal properties of the substrate while keeping the wetting properties the same (B. Sobac and D. Brutin, Phys. Rev. E, underpress). We experimentally investigate the behavior of a pinned droplet evaporating into air. The influences of the substrate temperature and substrate thermal properties on the evaporation process are studied in both hydrophilic and hydrophobic conditions. Experimental data are compared to the quasi-steady diffusion-driven evaporation model assuming the isothermia of the drop at the substrate temperature. This comparison permits to highlights several thermal mechanisms linked to evaporation and their respective contributions in regard of pure mass diffusion mechanism. The range of validity of the classical evaporation model is also discussed.

9:31AM A4.00008 Dropwise condensation on a cold gradient substrate[1] , ASHLEY MACNER, SUSAN DANIEL, PAUL STEEN, Cornell University — Distributions of drops that arise from dropwise condensation evolve by nucleation, growth, and coalescence of drops. An understanding of how surface-energy gradients applied to the substrate affect drop growth and coalescence is needed for design of effective surfaces for large-scale dropwise condensation. Transient dropwise condensation from a vapor phase onto a cold and chemically treated surface is reported. The surfaces were treated to deliver either a uniform contact-angle or a gradient of contact-angles by silanization. The time evolution of drop-size and number-density distributions is reported. For a typical condensation experiment, the drop distributions advance through two stages: an increase in drop density as a result of nucleation and a decrease in drop density as a result of larger scale coalescence events. Because the experiment is transient in nature, the shape of the distribution can be used to predict the number of drop generations and their stage of development. Preliminary results for gradient surfaces will be discussed and compared against observations of behavior on uniformly coated surfaces.

[1]NASA Space Technology Research Fellowship (NSTRF)

Sunday, November 18, 2012 8:00AM - 9:44AM –

Session A5 Computational Fluid Dynamics I 24A - Chair: Gianluca Iaccarino, Stanford University

8:00AM A5.00001 An Improved Advection Scheme for Implicit Interfaces on Cartesian Grids

, ZHIPENG QIN, University of Maryland, KEEGAN DELANEY, George Washington University, AMIR RIAZ, University of Maryland, ELIAS BALARAS, George Washington University — Existing methods for the advection of implicit interfaces on Cartesian grids employ a reconfiguration of the contours of the phase function in the neighborhood of the interface to facilitate the implementation of interfacial jump conditions. This is achieved with either the classical redistance techniques, within the context of the level set approach, or the more recent interface recompression methods that attempt to maintain interface width during advection. While the latter approach performs much better with respect to mass conservation we show that the associated interface topology suffers as a result. A new approach is presented to recreate interface topology in the neighborhood of the interface based on local curvature smoothing. With the help of canonical examples the approach is shown to both conserve mass and maintain topology during advection.

8:13AM A5.00002 The immersed interface method for simulating two-fluid flows[1]

, SHENG XU, MIGUEL UH, Southern Methodist University — In this talk, we present a second-order accurate implementation of the immersed interface method for computing a two-fluid flow. In the method, jump conditions in the flow fields caused by the effect of the two-fluid interface and the discontinuous fluid properties are incorporated into a numerical scheme on a Cartesian grid. We discuss how to derive and use these jump conditions to achieve second-order accuracy. We test the accuracy, efficiency and stability of our implementation on some canonical two-fluid flows.

[1]This work was supported by the NSF grant DMS 0915237.

8:26AM A5.00003 Interface-tracking using a compressive advection method and a compositional modelling approach[1]

, DIMITRIOS PAVLIDIS, JEFFERSON GOMES, CHRISTOPHER PAIN, OMAR MATAR, Imperial College London — We describe progress on a consistent approach for interface- tracking in which each component, representing a different phase/fluid, has a sum of unity. Our 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 that are assumed to be immiscible in this work although not a requirement of the approach. The method is based on a new mixed finite element pair, the P1DG-P2 element, in which pressure has a quadratic variation and velocity a discontinuous linear variation with the discontinuity between elements. This element allows us to represent key balances such as hydrostatic balance exactly assuming a linear variation in buoyancy. This means that on unstructured meshes we do not have problems representing these key balances that can result in large pressure gradients which, in turn, generate large spurious velocities that can dominate the solution. We apply the method to a series of benchmark problems that demonstrate the approach and show that the method works for at least three different fluids, and that it avoids putting a priority on resolving any of these fields or components.

[1]EPSRC Programme Grant EP/K003976/1

8:39AM A5.00004 Interface-tracking electro-hydrodynamic model for droplet coalescence

, LINDSAY CROWL ERICKSON, DAVID NOBLE, Sandia National Laboratories — Many fluid-based technologies rely on electrical fields to control the motion of droplets, e.g. micro-fluidic devices for high-speed droplet sorting, solution separation for chemical detectors, and purification of biodiesel fuel. Precise control over droplets is crucial to these applications. However, electric fields can induce complex and unpredictable fluid dynamics. Recent experiments (Ristenpart et al. 2009) have demonstrated that oppositely charged droplets bounce rather than coalesce in the presence of strong electric fields. Analytic hydrodynamic approximations for interfaces become invalid near coalescence, and therefore detailed numerical simulations are necessary. We present a conformal decomposition finite element (CDFEM) interface-tracking method for two-phase flow to demonstrate electro-coalescence. CDFEM is a sharp interface method that decomposes elements along fluid-fluid boundaries and uses a level set function to represent the interface. The electro-hydrodynamic equations solved allow for convection of charge and charge accumulation at the interface, both of which may be important factors for the pinch-off dynamics in this parameter regime.

8:52AM A5.00005 Numerical simulation in 3D of atomizing coaxial gas-liquid jets

, GILOU AGBAGLAH, DANIEL FUSTER, GEORDIE MCBAIN, Institut Jean Le Rond d'Alembert, UMR 7190, UPMC & CNRS, Paris, France, STEPHANE POPINET, NIWA, Kilbirnie, Wellington, New Zealand, STEPHANE ZALESKI, Institut Jean Le Rond d'Alembert, UMR 7190, UPMC & CNRS, Paris, France — We investigate three-dimensional multiphase flows using the Volume of Fluid method. We are in particular focusing on the problem of jet atomizaton. We use a Volume of Fluid method with oct-tree adaptive finite volume discretization, mostly using the Gerris free code. Surface tension is computed by a balanced-force method. Coaxial, 3D, round and planar air-water jets similar to those investigated experimentally are studied and compared to the equivalent jets in 2D axisymetric and 2D planar setups. A mechanism for large-scale jet disruption is observed. The distribution of droplet sizes is compared to experimental measurements. The effect of grid resolution and of the presence of an explicitly modelled solid separator plate is discussed.

9:05AM A5.00006 An Optimization-Based Lagrangian Particle Method for Navier-Stokes[1]

, PAUL COVINGTON, FRANK HAM, PARVIZ MOIN, CTR, Stanford University — Particle methods have recently experienced renewed interest due to their ability to more naturally handle material advection for multiphysics applications as well as complex moving boundaries. Most methods, however, lack a systematic treatment of formal accuracy. This study aims to provide a general framework for constructing Lagrangian methods that obey a bilinear form. Basis functions are defined independent of a traditional computational grid by borrowing concepts from convex optimization. The method is applied to the compressible Navier-Stokes equations and two-phase problems. Results will be presented with emphasis placed on problems with analytical solutions such as a convecting Euler vortex. Accuracy will be compared to an extensively validated Eulerian finite volume code. The applicability to 3D problems of practical interest will also be discussed with respect to algorithmic considerations and computational cost.

[1]Supported by Robert Bosch, LLC

9:18AM A5.00007 Particle Gradient-Augmented Level Set in Multiphase Flow Problems

, OLIVIER MERCIER, JEAN-CHRISTOPHE NAVE, McGill University, RODOLFO RUBEN ROSALES, MIT, BENJAMIN SEIBOLD, Temple University — The goal of this presentation is to present the particle gradient-augmented level set method in the context of multiphase flows. Specifically, we will show how topological changes can be taken into account by a modification of the original method. Finally, we will present applications to multi-phase Navier-Stokes by combining it with the ghost fluid method.

9:31AM A5.00008 Langrangian Coherent Structure Visualizations in High-Resolution DNS , DANIEL NELSON, San Diego State University — Lagrangian coherent structures have become an important addition to flow visualization and analysis of turbulent flow topology and separated flow. The identification of zero-mass flux material surfaces is shedding new light on flow transport phenomena. We discuss a high-order resolution determination of the Lagrangian coherent structures. In direct numerical simulation based on higher-order discontinuous Galerkin method, finite time Lyapunov exponents are determined on-the-fly in forward and backward-time based on spectral operators. A higher-order time integrator traces fluid particles that are initialized at quadrature points. Lagrangian coherent structures are visualized in a range of flows including a shear layer, convected vortex and bluff body wakes.

Sunday, November 18, 2012 8:00AM - 9:44AM –

Session A6 Electrokinetics I 24B - Chair: William Ristenpart, University of California, Davis

8:00AM A6.00001 Modeling the Transport of Colloids to Electrode Strips During Electrophoretic Deposition[1] , ANDREW PASCALL, KYLE SULLIVAN, JOSHUA KUNTZ, Lawrence Livermore National Laboratory — Electrophoretic deposition (EPD) is an industrially relevant process in which colloidal particles suspended in a liquid are forced to deposit on a electrode under an applied electric field. Studies of the formation of deposits by EPD have generally focused on electrode geometries that yield analytical solutions, such as infinite parallel planes and concentric cylinders. Here, we focus on an experimentally relevant geometry that has not yielded analytical solutions—the planar strip electrode. We present a finite element model for the transport of material onto a planar strip electrode which shows excellent qualitative agreement to experimental results in a similar system. Notably, we demonstrate that the presence of the edges of the electrode lead to a singularity in the electric field that significantly effects the morphology of the deposit at short times or for thin deposits.

[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. LLNL-ABS-567973.

8:13AM A6.00002 Electrohydrodynamic interactions of spherical particles under Quincke rotation , DEBASISH DAS, DAVID SAINTILLAN, Department of Mechanical Science and Engineering, University of Illinois, Urbana-Champaign — Quincke rotation denotes the spontaneous rotation of dielectric particles immersed in a slightly dielectric liquid when subjected to a high enough DC electric field. It occurs when the charge relaxation time of the particles is greater than that of the fluid medium, causing the particles to become polarized in a direction opposite to that of the electric field and therefore giving rise to an unstable equilibrium position. When slightly perturbed, the particles start to rotate, and if the electric field exceeds a critical value the perturbations do not decay and the particle rotations reach a steady state with a constant angular velocity. We use a combination of numerical simulations and asymptotic theory to study the effect of electrohydrodynamic interactions between particles under Quincke rotation. We study the prototypical case of two equally charged spheres carrying no net charge and interacting with each other both hydrodynamically and electrically. The case of spherical particles free to roll on a horizontal grounded electrode is also described. We show that Quincke rotation results in self-propulsion of the particles in the plane of the electrode, and interactions between a pair of such "rollers" are analyzed.

8:26AM A6.00003 Electro-orientation as a facile way to characterize the electrical properties of nanowires[1] , CEVAT AKIN, JINGANG YI, JERRY SHAN, Rutgers University, QI CHEN, WEIHE XU, YONG SHI, Stevens Institute of Technology — The electrical conductivity and/or permittivity of nanowires are often poorly known, heterogeneous, and difficult to measure by traditional direct-characterization methods such as 4-point-probes. Electro-orientation, the rotation of nanowires in liquid suspension into alignment with an external electric field, offers a potential alternative measurement technique that is simple and also compatible with further solution-based sorting and processing of nanowires. We present experimental results obtained by optical microscopy on the alignment rate of nanowires under spatially uniform AC electric fields of different frequency. Silicon nanowires of known conductivity were fabricated using metal-assisted chemical etching and tested to determine how the cross-over frequency for electro-orientation varies with particle conductivity and aspect ratio. We compare our experimental results with theoretically obtained values, and assess the potential of electro-orientation as a quantitative method of characterizing the electrical properties of large-aspect-ratio particles in liquid suspension.

[1]Supported by NSF CBET 0644719

8:39AM A6.00004 Accurate predictions of dielectrophoretic force and torque on particles with strong mutual field, particle, and wall interactions[1] , QIANLONG LIU, KENNETH REIFSNIDER, DOE Energy Frontier Research Center, University of South Carolina — The basis of dielectrophoresis (DEP) is the prediction of the force and torque on particles. The classical approach to the prediction is based on the effective moment method, which, however, is an approximate approach, assumes infinitesimal particles. Therefore, it is well-known that for finite-sized particles, the DEP approximation is inaccurate as the mutual field, particle, wall interactions become strong, a situation presently attracting extensive research for practical significant applications. In the present talk, we provide accurate calculations of the force and torque on the particles from first principles, by directly resolving the local geometry and properties and accurately accounting for the mutual interactions for finite-sized particles with both dielectric polarization and conduction in a sinusoidally steady-state electric field. Since the approach has a significant advantage, compared to other numerical methods, to efficiently simulate many closely packed particles, it provides an important, unique, and accurate technique to investigate complex DEP phenomena, for example heterogeneous mixtures containing particle chains, nanoparticle assembly, biological cells, non-spherical effects, etc.

[1]This study was supported by the Department of Energy under funding for an EFRC (the HeteroFoaM Center), grant no. DE-SC0001061.

8:52AM A6.00005 Role of Solution Conductivity in Reaction Induced Charge Auto-Electrophoresis , JEFFREY MORAN, JONATHAN POSNER, University of Washington — The self-propelled motion of bimetallic particles in hydrogen peroxide solutions has been widely investigated. Multiple studies have predicted or reported that the swimming speed of these particles scales inversely with solution conductivity. We use scaling analyses and simulations to investigate the physical mechanism for the conductivity-induced deceleration. In particular, we focus on the interaction between dipolar charge density in the fluid surrounding the rod and the electric field it generates, which is weakened by the addition of electrolyte. The simulations show good agreement both with experimental data and with previous analytical treatments of the conductivity dependence.

9:05AM A6.00006 Reservoir-based dielectrophoresis (rDEP) for continuous separation of particles based upon surface charge , XIANGCHUN XUAN, SAURIN PATEL, Clemson University, SHIZHI QIAN, Old Dominion University — Separating particles from complex mixtures is important to many applications. We develop a continuous-flow microfluidic approach to separating 3 μm fluorescent and non-fluorescent particles by charge inside a reservoir under DC-biased AC electric fields. This separation exploits the reservoir-based dielectrophoresis (rDEP), which is induced by the inherent electric field gradient formed at the reservoir-microchannel junction, to continuously isolate the trapped fluorescent particles from the streaming non-fluorescent particles. The obtained particle images agree closely with the predicted particle trajectories from a 2D numerical model. It is, however, found that the streaming non-fluorescent particles may also get trapped in the reservoir due to the influences from the accumulated fluorescent particles, which can significantly lower the separation purity. These influences decrease with the enhanced electrokinetic flow (by increasing the applied DC electric field) and the lowered AC field frequency. Since it takes place inside the reservoir and no in-channel mechanical or electrical parts are needed, the demonstrated rDEP particle sorter can be conveniently integrated with other components into lab-on-a-chip devices for diverse particle handling.

9:18AM A6.00007 The Effect of Electrophoresis and Electroosmosis on Colloid Dynamics at a Micro-Nano-Channel Junction , GILAD YOSSIFON, YOAV GREEN, Technion - Israel Institute of Technology — Understanding the electrokinetic interaction between nano-colliodal particles and a nano-channel is of particular interest in the fast growing field of micro- and nano-fluidics. It is well established that both nanocolloids and nanochannels/nanopores play an important role in biomolecular detection. Combining these may open new routes for a more sensitive detection platform. Our design consists of a nano-slot bounded by two micro-chambers, wherein we introduce the dispersed nano-colloids. We drive the fluid and particles into the channel via an electric field using electrophoresis and electro-osmosis. We have derived an analytic expression for the colloid dynamics within the microchamber, away from the nanoslot entrance vicinity where dielectrophoresis effects are dominant. We account for the two opposing mechanisms - electrophoresis and electroosmosis. The latter accounts for the dependence of nanochannel electro-osmotic permeability on electric Debye layer overlap intensity. These theoretical results stand in good qualitative agreement with the experimental findings.

9:31AM A6.00008 Rapid annealing of polycrystalline domains with a hexatic-to-disorder transition in colloidal crystals near electrodes , C.S. DUTCHER, N.H. TALKEN, T.J. WOEHL, W.D. RISTENPART, Dept. Chemical Engineering and Materials Science, Univ. 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, suggesting that the driving force for crystallization should increase at lower frequencies. 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$1000 Hz), spherical micron-scale particles form HCP crystals; as the frequency is decreased below $\sim$250 Hz, however, the crystalline structure transitions to randomly close packed (RCP) crystals. 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 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.

Sunday, November 18, 2012 8:00AM - 9:31AM –

Session A7 Microfluidics: Methods and Devices I 24C - Chair: Amir Hirsa, Rensselaer Polytechnic Institute

8:00AM A7.00001 Microfluidic synthesis of crimped fibers , JANINE NUNES, Princeton University, HANNAH CONSTANTIN, Yale University, TALAL AL-HOUSSEINY, HOWARD STONE, Princeton University — Flexible high aspect ratio microstructures, such as microfibers, are of considerable interest for potential textile, rheological and life science applications. This research is focused on the microfluidic synthesis of wavy or crimped polymeric microfibers. It is known that highly viscous liquid threads sheathed by a low viscosity continuous phase liquid can buckle when allowed to flow through a microchannel where there is an increase in channel cross-sectional dimensions. These structures are transient and evolve during flow to form piles, relax to straight threads or coalesce. Here we present the first example where buckling is triggered by the initiation of a polymerization reaction in a liquid thread that does not initially exhibit buckling (because of the low viscosity ratio between thread and continuous phase), and the subsequent preservation of the buckled morphology through completion of the crosslinking reaction. The resulting microfibers have highly uniform and reproducible morphology. By changing the location in the channel where the reaction is initiated, as well as the flow rates, the degree of waviness of the microfibers can be controlled. Current efforts are focused on developing a physical understanding of this process.

8:13AM A7.00002 The use of sequences of pillars to engineer fluid cross-sectional shape *via* inertial flow deformations , HAMED AMINI, MAHDOKHT MASAELI, ELODIE SOLLIER, UCLA, YU XIE, BASKAR GANAPATHYSUBRAMANIAN, Iowa State University, HOWARD A. STONE, Princeton University, DINO DI CARLO, UCLA — Control of fluid streams is useful in biological processing, chemical reaction engineering, and creating structured materials. We use cylindrical pillars to induce significant deformations in laminar flow. Numerical simulations predict that as fluid passes centrally positioned pillars in a straight microchannel, the fluid parcels near the channel centerline move towards the side walls, while fluid parcels near the top and bottom walls move towards the center. This inertial phenomenon ($1 \lesssim Re \lesssim 100$) in effect creates a set of *net* rotational secondary flows within the channel. The existence of four dominant operating modes (based on the number and direction of the induced secondary flows) is also demonstrated. We show how using the basic deformations on miscible co-flows of water and fluorescent dye we can manipulate and shape the cross-section of the colored stream. Hierarchical flow deformation operations can be integrated to execute sophisticated programs and render complex flow-shapes. We can numerically predict the deformation near a single pillar with high precision and accordingly, predict the total transformation function of any potential program. Consequently, a user can use a library of pre-simulated motions and engineer a flow-shape of interest quickly, at a low cost, and with high accuracy, an ability which we demonstrate.

8:26AM A7.00003 Using in-fiber fluid instabilities for the scalable production of structured spherical particles , AYMAN ABOURADDY, JOSHUA KAUFMAN, GUANGMING TAO, SOROUSH SHABAHANG, ESMAEIL-HOOMAN BANAEI, University of Central Florida, DAOSHENG DENG, XIANGDONG LIANG, STEVEN JOHNSON, YOEL FINK, Massachusetts Institute of Technology — Developing efficient pathways to the fabrication of spherical particles is needed for a wide range of applications ranging from drug delivery to cosmetics. A heretofore unanswered technological challenge is the development of a single process for fabricating particles over a wide range of sizes, from a variety of materials, and in different structures. Here we harness the high-volume process of fiber drawing to create a scalable nanomanufacturing approach to address this challenge. We make use of a new class of multi-material fibers drawn from a macroscopic "preform" combined with the recent observation of an in-fiber Plateau-Rayleigh capillary instability (PRI). A macroscopic cylindrical preform is drawn into an extended fiber and subsequent thermal treatment induces the PRI at the heterogeneous interfaces along the fiber, causing the core to break up into a necklace of uniformly sized spheres held stationary in isolation in a cladding matrix. This process enables the fabrication of structured spherical particles from a variety of materials spanning an unprecedented range of sizes: from 2 mm down to 20 nm. By structuring the core at the preform stage, we produce multi-material core-shell particles, "Janus" particles, and multi-sectioned "beach ball" particles. By combining multiple cores in the same fiber we demonstrate an exceptionally high level of parallelization, rendering the process scalable.

8:39AM A7.00004 Flow-induced protein crystallization: Macroscopic effects on 2D crystals , JAMES YOUNG, DAVID POSADA, AMIR HIRSA, Rensselaer Polytechnic Institute, JUAN LOPEZ, Arizona State Univeristy — Proteins must first be crystallized before their molecular structure can be studied in detail. However, crystallizing protein is a challenging task which is often met with limited success. Although 2-D protein crystals at the air/water interface are usually obtained under quiescent conditions, it was recently shown that crystallization can be enhanced by a shearing flow. Here we examine the relationship between Reynolds number and the crystal growth process using the deep-channel surface viscometer geometry. It consists of an annular region bounded by stationary inner and outer cylinders and driven by a constant rotation of the floor. The interfacial velocity measurements are compared to Navier-Stokes computations with the Boussinesq-Scriven surface model. The interfacial film is lifted onto a solid substrate, and the protein crystals are observed via optical and atomic force microscopy. For a particular protein surface concentration, a Reynolds number threshold has been identified for flow-induced crystallization. This flow geometry also allows for the determination of the surface shear viscosity, which provides a quantitative measure of the mesoscale interactions associated with protein crystallization.

8:52AM A7.00005 The Correlated Dynamics of a Pair of Tethered Microcantilevers in a Viscous Fluid[1] , BRIAN ROBBINS, MILAD RADIOM, JOHN WALZ, WILLIAM DUCKER, MARK PAUL, Virginia Tech — Understanding the dynamics of biomolecules or polymers in a fluid environment is an important challenge. One approach is to tether a molecule between the ends of two Brownian driven microcantilevers and to measure the change in their correlated dynamics. However, the cantilever dynamics is also correlated due to the motion of the intervening viscous fluid. An important question is whether the correlations due to a tethered molecule can be measured in the presence of the fluid coupling for configurations accessible to experiment. We present experimental measurements of the correlated motion of two microcantilevers in water without a tethered molecule. Using the fluctuation-dissipation theorem with deterministic finite-element simulations we compute the correlated dynamics for laboratory conditions. Our numerical results show very good agreement with experimental measurement. We next include a linear spring between the cantilever tips to model a tethered molecule and quantify the dynamics of the cantilever pair for a wide range of conditions. Our results provide physical insights into the signature of a tethered molecule and quantify the force, time, and length scales that are accessible to current technologies.

[1]NSF Award CBET-0959228

9:05AM A7.00006 Measuring ultralow interfacial tensions with magnetic particles in microchannels , SCOTT TSAI, University of Toronto, JASON WEXLER, Princeton University, JIANDI WAN, Rochester Institute of Technology, HOWARD STONE, Princeton University — Ultralow interfacial tension solutions have interfacial tensions 1,000 times, or more, smaller than typical oil-water solutions. We describe a technique that measures ultralow interfacial tensions by magnetically deflecting paramagnetic spheres in a co-flow microfluidic device. Our method involves tuning of the distance between the co-flowing interface and the magnetic field source, and observing the behavior of the magnetic particles as they approach the liquid-liquid interface – the particles either pass through or are trapped. We demonstrate the effectiveness of this technique for measuring very low interfacial tensions by testing solutions of different surfactant concentrations, and we show that our results are comparable with measurements made using a spinning drop tensiometer.

9:18AM A7.00007 Ferrodroplets: a Global Clock for Droplet Microfluidics , GEORGIOS KATSIKIS, MANU PRAKASH, Stanford University — A fluidic analogue to magnetic bubble computer memory is proposed as a novel propagation mechanism for droplet microfluidics. We designed a prototype microfluidic device where ferrofluid droplets are actuated along a 2-D plane using soft magnet patterns under the influence of rotating magnetic fields. The state of the system is dependent on occupancy of fluid droplets and the track geometry. The propagation characteristics of droplets are studied experimentally by varying operation parameters such as the magnitude and frequency of in-plane magnetic fields and the length scale of the device. The experimental findings are juxtaposed with scaling arguments and numerical simulations. Applications for this device as a universal clocking mechanism for droplet microfluidics are discussed.

Sunday, November 18, 2012 8:00AM - 9:44AM –

Session A8 Particles: Gravity and Settling 25A - Chair: Kenneth Kiger, University of Maryland

8:00AM A8.00001 Settling Regimes of Inertial Particles in Turbulence[1] , GARRETT GOOD, Sibley School of Mechanical and Aerospace Engineering at Cornell University (CU-MAE), Max Planck Institute for Dynamics and Self-Organisation (MPI-DS), PETER IRELAND, CU-MAE, EWE WEI SAW, GREGORY BEWLEY, MPI-DS, EBERHARD BODENSCHATZ, CU-Physics, MPI-DS, ZELLMAN WARHAFT, CU-MAE — We present numerical and experimental evidence regarding the (1) enhancement and (2) reduction of particle settling speeds from their Stokes velocities by turbulence. Settling enhancement has long been attributed to particle path biases while various models have been proposed for settling reduction, including vortex trapping, loitering, and most commonly, non-linear drag effects. There is disagreement between experiment and DNS, and this work in particular is motivated by that highlighted in the entrainment study of Ireland & Collins (2012). Here, we take a new look at particles with both enhanced and reduced settling velocities, and focus on the transition between these two regimes and how it is affected by the large- and small-scale features of the turbulence. The numerical results show the relevant parameter space in unprecedented detail, while the experiments use a new apparatus which creates uniquely adjustable turbulence at its center (generated by thirty-two randomly driven loudspeaker jets) to measure settling speeds in real flows. We address current contention, and aim to paint a more comprehensive picture of particle settling.

[1]Fulbright Program, Max Planck Society, National Science Foundation

8:13AM A8.00002 Dynamics of gravity-driven, particle-laden thin-film flows , ALIKI MAVROMOUSTAKI, ANDREA BERTOZZI, UCLA — Our study focusses on gravity-driven, particle-laden flows which are pertinent to a wide range of industrial and geophysical settings in which transport of suspensions occur. In the case of gravity-driven, single-species, particle-laden flows, there exist three distinct regimes which are dependent on the plane angle of inclination and bulk particle volume fraction: settling of particles to the substrate versus settling to the front of the flow and, an intermediate, unstable, "well-mixed" regime. The dynamics is described by a previously derived equilibrium model, using lubrication theory, based on a balance between hindered settling and shear-induced migration; this consists of a coupled system of hyperbolic conservation equations which describe the interface position and the particle volume fraction. We investigate the governing system analytically and numerically; an analysis of the governing equations exhibits rich mathematical structure where we observe the formation of double-shock wave solutions while, as the limit of maximum permissible particle concentration is approached, the numerical solutions are described by singular shocks. Finally, we discuss the physical interpretation of our solutions as applied to the experimental setting.

8:26AM A8.00003 Dynamics of particle settling and resuspension in viscous liquid films[1] , ANDREA BERTOZZI, Department of Mathematics, University of California Los Angeles, NEBOJSA MURISIC, Princeton University, BENOIT PAUSADER, Courant Institute of Mathematical Sciences, New York University, DIRK PESCHKA, Weierstrass-Institute for Applied Analysis and Stochastics, Berlin — We develop a dynamic model for suspensions of negatively buoyant particles on an incline. Our model includes settling due to gravity as well as resuspension of particles by shear-induced migration. We consider the case where the particles settle onto the solid substrate and two distinct fronts form, namely a faster liquid and a slower particle front. We show that the resulting transport equations for the liquid and the particles are of hyperbolic type, and study the dilute limit, for which we compute exact solutions. We also carry out systematic laboratory experiments, focusing on the motion of the liquid and the particle fronts. We show that the dynamic model predictions for small to moderate values of the particle volume fraction and the inclination angle of the solid substrate agree well with the experimental data.

[1]This work was supported by NSF grant DMS-1048840 and UC Lab Fees 976 Research Grant 09-LR-04-116741-BERA.

8:39AM A8.00004 Effect of gravity on the preferential concentration of heavy particles[1] , YONG-NAM PARK, CHANGHOON LEE, Yonsei University — The effects of gravity on the preferential concentration of heavy particles are investigated in forced isotropic turbulence using direct numerical simulation. Preferential concentration of heavy particles is usually observed for particles with St$\approx$ 1 and the mechanism of preferential concentration is strongly related with the vortical structure of background flows. In this study we found that strong gravity causes a different kind of preferential concentration for high Stokes number particles which is not related with the vortical motion of fluid. We introduce average distance concept between particles for quantitative analysis of preferential concentration, and suggest that the closest distance between particles is a good indicator of preferential concentration. Moreover, we investigate the effect of gravity on the geometric nature of heavy particle's trajectories such as curvature and torsion. PDF of curvature and torsion are determined by the Gaussian distribution of particle velocity, and thus their PDFs are not modified by gravity as long as the particle's velocity maintains the Gaussian characteristics. More detailed analysis will be discussed in the meeting.

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

8:52AM A8.00005 A Voronoi Analysis of Preferential Concentration of Heavy Particles in Active Grid Generated Turbulence , MARTIN OBLIGADO, ALAIN CARTELLIER, MICKAEL BOURGOIN, Laboratoire des Ecoulements Geophysiques et Industriels, CNRS/UJF/G-INP UMR 5519, BP53, 38041, Grenoble, France — Particle laden flows are of relevant interest in many industrial and natural systems. When the carrier flow is turbulent, a striking feature is the phenomenon called preferential concentration: particles denser than the fluid have the tendency to inhomogeneously distribute in space, forming clusters and depleted regions. We present a study on the preferential concentration and clustering in homogeneous and isotropic turbulence based on Voronoi diagrams. We have formerly quantified preferential concentration as a function of the Stokes number (defined as the ratio of the particle viscous relaxation time to dissipation timescale of the flow) in moderate turbulence conditions, up to Reynolds number based on Taylor microscale of the order of $R_\lambda \sim 120$. Using an active grid recently implemented in our windtunnel, we investigate in the present study the effect of Reynolds number on particles clustering in the range $R_\lambda \sim 200 - 400$. Clustering level is found to be significantly higher than previous measurements at lower Reynolds number. We also present an analysis of the geometry of clusters and voids and investigate the possible connection with stick-sweep mechanisms using direct numerical simulation data of homogeneous isotropic turbulence.

9:05AM A8.00006 How gravity and size affect the acceleration statistics of bubbles in turbulence , VIVEK N. PRAKASH, YOSHIYUKI TAGAWA, Physics of Fluids Group, University of Twente, The Netherlands, ENRICO CALZAVARINI, Laboratoire de Mecanique de Lille CNRS/UMR, Universite Lille 1, France, JULIAN MARTINEZ MERCADO, Physics of Fluids Group, University of Twente, The Netherlands, FEDERICO TOSCHI, Department of Physics, and Department of Mathematics and Computer Science, Eindhoven University of Technology, The Netherlands, DETLEF LOHSE, CHAO SUN, Physics of Fluids Group, University of Twente, The Netherlands, INTERNATIONAL COLLABORATION FOR TURBULENCE RESEARCH COLLABORATION — We report results from a Lagrangian experimental investigation in the largely unexplored regime of very light (air bubbles in water) and large particles ($D/\eta >> 1$) in turbulence. Using a traversing camera setup and particle tracking, we study the acceleration statistics of $\sim 3\ mm$ diameter (D) bubbles in a water tunnel with nearly homogeneous and isotropic turbulence generated by an active-grid. The experiments reveal that gravity increases the acceleration variance and reduces the intermittency of the PDF in the vertical direction. Moreover, the experimental acceleration PDF shows a substantial reduction in intermittency at growing size ratios, in contrast to neutrally buoyant or heavy particles. All these results are closely matched by numerical simulations of finite-size bubbles with Faxén corrections.

9:18AM A8.00007 Modeling Gravitational Settling of Inertial Particles in Turbulent Like Flow , SATHYANARAYANA AYYALASOMAYAJULA, SOHAM BANERJEE, Indian Institute of Technology Bhubaneswar, ZELLMAN WARHAFT, Cornell University — Gravitational settling of inertial particles in a turbulent fluid is an important aspect of many natural and engineering flows. The recent experiments of Good et al 2012, *J. Fluid Mech*, 694, and earlier experiments reveal the presence of two effects, loitering and fast-tracking that modulate the gravitational settling rates of the inertial particles in a turbulent flow. The equivalent DNS simulations of Ireland & Collins 2012, *J. Fluid Mech*, 704, in many respects show good agreement with the experiments but the magnitude of loitering and fast-tracking is much smaller. The Vortex model of Ayyalasomayajula et al 2008, *Phys. Fluids*, 095104, was earlier used to study the inertial particle acceleration statistics in turbulent like model flow to explain the reduced inertial particle accelerations. The Vortex model is again used here in this current study to better understand the nature of fast-tracking and loitering effects for a turbulent like fluid flow. We present results where the parametric variation of fluid flow intensity (analogous to changing Re), Stokes numbers and gravity is performed and the relative changes of loitering and fast-tracking effects are studied. We offer possible explanations for the discrepancy between the experiments and DNS also.

9:31AM A8.00008 Effect of ambient flow inhomogeneity on shear-induced lift on a sphere at finite Reynolds number , JUNGWOO KIM, Seoul National University of Science and Technology — In 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 based on uniform or other simple linear ambient flows such as uniform shear and uniform vortex. Therefore, the objective of this study is to investigate the effect of flow inhomogeneity on shear-induced lift on a sphere. To do so, we perform direct numerical simulations of a sphere in an inhomogeneous shear. One of the inhomogeneous shear flows considered is the sine profile having the form of $u(x,y)/U_\infty = 1 + K\sin(2\pi\frac{y/D}{L/D})$. Here, U_∞ is the fluid velocity at the center of the sphere, D the sphere diameter, L the period of the sine profile. Also, the Reynolds number is $\frac{U_\infty D}{\nu} = 100$. The present simulations show that the lift forces are decreased with increasing the degree of the flow inhomogeneity (that is, D/L) while the non-dimensional shear rate at the location of sphere center is fixed to be $2\pi\frac{K}{L} = 0.1$. Comparing the change in the lift force with respect to surface-averaged vorticity under uniform inlet shear, that under inhomogeneous shear has certain systematic deviations. In the final presentation, more details of the shear-induced lift on a sphere in inhomogeneous shear flows considered would be presented.

Sunday, November 18, 2012 8:00AM - 9:44AM –

Session A9 Interfacial/Thin Film Instability I 25B - Chair: Thomas Ward, North Carolina State University

8:00AM A9.00001 Gas-driven displacement of a non-Newtonian liquid in a radial Hele-Shaw cell , ANDREW WHITE, THOMAS WARD, Iowa State University, Department of Aerospace Engineering — The displacement of a non-Newtonian liquid by a less viscous fluid has applications in a number of industries such as lubricating oils, injection molding and cement placement in oil wells to name a few. A convenient geometry to study such a problem is that of the Hele-Shaw cell due to its ability to effectively reduce the flow to two dimensions when the gap spacing is much smaller than the other spatial dimensions. We will study the radial displacement of a finite drop of non-Newtonian shear-thinning and extensionally-thickening liquid by a gas at constant pressure in a Hele-Shaw cell with gap spacing O(10-100) microns. Different concentrations of a polymer in oil will be used to examine changes in the displacement rate, residual film thickness and resulting Saffman-Taylor instability as the viscoelastic time scale overtakes that of the bulk displacement.

8:13AM A9.00002 Wave Structure and Velocity Profiles in Downwards Gas-Liquid Annular Flows , IVAN ZADRAZIL, GEOFF HEWITT, OMAR MATAR, CHRISTOS MARKIDES, Imperial College London — A downwards flow of gas in the core of a vertical pipe, and of liquid in the annulus between the pipe wall and the gas phase is referred to as a "downwards annular flow" (DAF). DAFs are conventionally described in terms of short-lived, small-amplitude "ripples," and large-amplitude, high-speed "disturbances." We use a combination of Laser Induced Fluorescence (LIF), Particle Image and Tracking Velocimetry (PIV, PTV) to study DAFs. We demonstrate through these techniques that the liquid films become progressively more complex with increasing liquid Reynolds number (ReL), while a similar increase of complexity is observed for increasing gas Reynolds number (ReG). Disturbance waves are observed for low and high ReL, and ripples for intermediate ReL. Additionally, a high degree of rolling breakdown of disturbance waves is observed in falling films at the highest ReL, which is a source of bubble entrainment into the film body. Our results will comprise: (i) statistical data on film thickness, and (ii) wave frequency, velocity, wavelength. In addition, a qualitative (e.g. re-circulation zones) and quantitative (e.g. mean/rms velocity profiles) velocity characterisation of the film flows will be presented.

8:26AM A9.00003 Wave structure in Upwards Gas-Liquid Annular Flows , YUJIE ZHAO, GEOFF HEWITT, OMAR MATAR, CHRISTOS MARKIDES, Imperial College London — A two-phase flow system in a vertical pipe in which the liquid around the pipe periphery is lifted by the gas core is referred to as an "upwards annular flow" (UAF). UAFs have a complex interfacial structure, which consists of short-lived, small-amplitude "ripple" waves, and large amplitude, high-speed "disturbances" waves. Two sets of flush-mounted electrically conducting probes together with axial view photography were used to study UAFs. The overall wave frequency decreased with increasing distance from the inlet until saturation. Disturbance waves were observed over a wide range (both low and high) of liquid Reynolds numbers, ReL, while ripples were observed at lower ReL. Disturbance "bursts," which are a source of liquid entrainment into the gas core, were also observed, with increasing frequency at progressively higher ReL. The waves appeared more chaotic near the inlet, which hindered the formation of the correlated waves. As the small (ripple) waves coalesced into bigger waves with increasing distance from the inlet, the waves became more coherent around the pipe periphery. The results that will be presented comprise: (i) statistical film thickness data, and (ii) wave, frequency, velocity, and wavelength.

8:39AM A9.00004 Viscous liquid thin-film flow inside a tube[1] , H. REED OGROSKY, ROBERTO CAMASSA, M.G. FOREST, CHRIS JOY, JEEHO KIM, JEFFREY OLANDER, University of North Carolina — Experiments are conducted over a range of parameters where a high-viscosity silicone oil is fed into the top of a vertical thin glass tube. The oil flows at a continuous rate and the resulting gravity-driven flow coats the inside of the tube. Interfacial instabilities develop due to surface tension and azimuthal curvature. Depending on the experimental parameters, the instabilities either saturate and propagate down the tube as traveling waves or form propagating plugs or liquid bridges. Using a long-wave asymptotic model, we compare the growth rates and phase speed predicted by linear stability analysis with those of the experiments, and the fully nonlinear form of the model is used to predict the formation of plugs. Comparison of the model and experiments to its counterpart exterior setup of a gravity-driven film flow coating a fiber will be mentioned. The experiments are then extended to include pressure-driven airflow at a constant flow rate upwards through the center of the tube. The interfacial stress created by the airflow alters the speed and growth rates of the instabilities. The leading-order effects of the airflow are included in the long-wave model, and a comparison is made once again between model and experiments.

[1]We gratefully acknowledge support from NSF RTG DMS-0943851 and NIEHS 534197-3411.

8:52AM A9.00005 Reconstruction of a Slippery Undulated bottom Substrate for a thin film flow over it with a Prescribed Spatially Periodic Free Surface , USHA RANGANATHAN, Professor, Department of Mathematics, Indian Institute of Technology Madras — A gravity-driven film flow on a slippery undulated inclined substrate is considered. The inverse problem of finding the topography of the bottom slippery substrate for a specific free surface shape which is evolved for flow over it is investigated. Applying the Energy Integral Method, evolution equations for the flow rate and the film thickness are obtained. The influence of slip coefficient, film thickness, inertia and surface tension on the shape of the slippery substrate is examined by prescribing the free surface of the flow over it as a mono frequent periodic function. The contour of the slippery undulated bottom substrate is obtained analytically for weakly undulated free surface. The results of the numerical simulations reveal that the slippery bottom substrate may become strongly nonlinear. The linear stability of the corresponding direct problem is examined and the critical Reynolds number for the flow with a fixed undulated free surface is found to be strongly influenced by surface tension.

9:05AM A9.00006 Dynamics of turbulent falling films , LENNON O'NARAIGH, University College Dublin, OMAR MATAR, Imperial College London — The dynamics of laminar falling films have received considerable attention over the past several decades. In contrast, turbulent falling films have been the subject of far fewer studies. We seek to redress this balance by studying the stability of falling films which have already undergone a transition from a laminar to a turbulent flow regime. We derive a uniform-film base-state for this flow by assuming the averaged turbulent velocity field to be steady and fully-developed, and by employing a modified version of mixing-length theory. The latter features an interpolation function for the eddy viscosity, and van Driest-type functions for turbulence-damping near the wall and interface regions. The predicted base-state streamwise velocity component is in good agreement with experimental data. A linear stability analysis of this base-state is then carried out by solving a modified version of the Orr-Sommerfeld equation. Our results suggest that the unstable mode is a long-wave one. This provides motivation for the derivation of long-wave equations for the nonlinear evolution of the film.

9:18AM A9.00007 Linear stability of a liquid film on a slipping surface with nonuniform slip length , ELIZAVETA GATAPOVA, Institute of Thermophysics, Novosibirsk (Russia), VLADIMIR AJAEV, Southern Methodist University, OLEG KABOV, Institute of Thermophysics, Novosibirsk (Russia) — We consider stability of a uniform liquid film on a structured surface. The effect of structuring is modeled by periodic spatial variations of slip length, motivated by a situation when gas-liquid or liquid-liquid menisci form between the elements of the structure. The film instability is driven by disjoining pressure which is modeled by body force terms in the Navier-Stokes equations. Previous investigations of this configuration were based on lubrication-type models, which fail when localized regions of rapid change in surface properties are present, e.g. near the lines of contact between fluid menisci and solid surface. We develop a linear stability theory which is free from the limitations of the lubrication-type approach. Destabilizing effect of the surface structuring is found. The results are compared with the case of uniformly slipping surface.

9:31AM A9.00008 On the instability of a circular hydraulic jump , HAMID AIT ABDERRAHMANE, ASLAN KASIMOV, King Abdullah University of Science and Technology (KAUST) — We present results of an experimental investigation of symmetry breaking of a circular hydraulic jump observed when a vertical jet of fluid impinges on a horizontal plate. Instabilities break the axial symmetry of the circular jump into quasi-steady polygonal patterns. In between, there exist irregular and unsteady asymmetric jumps. The dynamics of these patterns is recorded experimentally and analysed subsequently by Fourier spectral methods. The attractors that depict the dynamics of are reconstructed and analyzed with the aid of delay and nonlinear time series analysis methods.

Sunday, November 18, 2012 8:00AM - 9:44AM –

Session A10 Instability: Jets, Wakes and Shear Layers I 25C - Chair: Colm-cille Caulfield, BP Institute and DAMTP Cambridge

8:00AM A10.00001 Nonlinear destabilization of stably stratified shear flow[1] , NADIA MKHININI, THOMAS DUBOS, PHILIPPE DROBINSKI, Laboratoire de meteorologie dynamique, Ecole polytechnique/IPSL — The supercritical or subcritical nature of the bifurcation occurring at a critical bulk Richardson Ric in stratified shear flows is investigated. This investigation is motivated by the recent observation that, in the stratified Ekman boundary layer, both supercritical and subcritical bifurcations occur for low and high values of the Prandtl number Pr, respectively. Linear stability of stratified shear flows is well-described by the Miles-Howard criterion, but the nature of bifurcation is determined by stabilizing or destabilizing nonlinear feedback mechanisms. To identify such mechanisms, an amplitude equation is derived near Ric. The variations of the first Landau coefficient mu as a function of Reynolds number, Prandtl number and wave vector of the shear instability are studied. Pr is the leading factor determining the sign of mu and the nature of bifurcation. The underlying mechanism is that vertical mixing induced by shear instability reduces background gradients of both velocity and buoyancy. The shear-induced feedback is stabilizing while the stratification-induced feedback is destabilizing and stronger when diffusion is low and Pr is high. The weakly nonlinear analysis is repeated on the continuously stratified Kelvin-Helmholtz flow with identical conclusions.

[1]This work was supported by the INSU/LEFE program and by the Agence Nationale de la Recherche, contract ANR-09-JCJC-0108-01.

8:13AM A10.00002 On the dynamics of shear layers formed on the interface between a porous strip and a clear fluid , PANAGIOTIS-DIMITRIOS ANTONIADIS, MILTIADIS V. PAPALEXANDRIS, Universite catholique de Louvain — In this talk we present results from 2D and 3D simulations of a temporally-evolving shear layer that is developed on the interface between a porous strip of large porosity and a clear fluid. The simulations are based on a single set of governing equations, valid for both inside and outside the porous layer, that does not require additional conditions on the interface. These equations are integrated via a predictor-corrector, projection-based scheme on a collocated grid. According to our study, the evolution of the shear layer can be divided in 4 phases. The first one is characterised by the onset of the Kelvin-Helmholtz instability, whereas in the second, the layer's momentum thickness grows according to the square-root of time law. The third phase is marked by roll-up and formation of vortices that extend to the interior of the porous medium; nonetheless the spatially-averaged velocities remain self-similar. In the fourth phase, the growth rate is much higher and the flow eventually experiences a transition to turbulence. Our talk concludes with results from a parametric study with respect to the porosity of the porous strip.

8:26AM A10.00003 Oblique laminar-turbulent interfaces in plane shear flows , YOHANN DUGUET, LIMSI-CNRS, PHILIPP SCHLATTER, KTH-Linné Flow Centre — In many wall-bounded shear flows, turbulence can spread in the presence of finite-amplitude perturbations despite the linear stability of the base flow. The onset of the transitional regime is usually characterised by the formation of large-scale oblique patterns of alternatively laminar and turbulent flow. Yet the mechanism responsible for the observed obliqueness has so far remained mysterious. In this talk we will focus on the formation of such oblique structures in plane Couette flow, using both analytical arguments and intensive direct numerical simulations. We will suggest a robust mechanism for the obliqueness of the incipient turbulent spots derived from mass and momentum budgets in the regions close to the laminar/turbulent interfaces.

8:39AM A10.00004 Gravity, surfactants and interfacial instabilities of shear flows , DAVID HALPERN, ALEXANDER FRENKEL, ADAM SCHWEIGER, University of Alabama — We study the linear-stability properties of slow two-fluid plane Couette-type flows in the presence of gravity and surfactants. If gravity is absent, the flow is unstable in certain regions of parameter space due to insoluble surfactants, while in other parametric regions, surfactants are stabilizing; in the absence of surfactants, gravity may lead to the Rayleigh-Taylor instability while it is stabilizing if the lighter liquid is the top layer. Due to the surfactant, there are two active normal modes, and thus two dispersion curves. For small enough Marangoni numbers Ma, the instability, if any, is longwave at its onset (reported earlier). At larger Ma, the instability close to its onset may be "midwave," where the growth rate is positive for a finite interval of nonzero wavenumbers. We present arbitrary-wavenumber results that involve the Bond number, Ma, the velocity-shear, the viscosity ratio and the aspect ratio of the two layers. We also present results for the special limit of infinite aspect ratio. There are dispersion curves with two maxima - a result of the crossing and reconnection of the dispersion curves as Ma or another parameter varies. Also, as Ma increases, for fixed values of the other parameters, the flow instability may switch on and off multiple times.

8:52AM A10.00005 Trailing edge effect on fast mixing in forced confined mixing layers , WEI ZHAO, GUIREN WANG, University of South Carolina — It was believed that due to nonlinear effect and saturation, the spreading rate in forced 2D mixing layers can only reach about two times of that in the unforced one. The limited enhancement restricted the related technique in practical application. We found recently that confined mixing layer can overcome the saturation under a specific and narrow frequency band to achieve ultra fast mixing. Here, we report the spanwise vortices are extremely sensitive to the sharpness of the trailing edge. Without trailing edge, there are no spanwise vortices. In a blunt trailing edge, there can be spanwise wortices, but there is no spanwise counter-rotation vortice. With sharp trailing edge, there are large spanwise vortices and at high forcing level counter-rotation vortices, which can cause initially fast mixing. The influence of trailing edge sharpness on this spanwise counter-rotation shredding vortices could be explained by the relation between acoustic particle displacement (APD) and the curvature radius of trailing edge. When the APD becomes larger or equal to the radius of curvature, the acoustically induced shredding vortices emerge due to the friction of wall and nonlinear flow. That could be why sharper trailing edge can induce strong vortices and fast mixing.

9:05AM A10.00006 Transient perturbation growth in time-dependent mixing layers , C.P. CAULFIELD, BP Institute & DAMTP, University of Cambridge, CRISTOBAL ARRATIA, JEAN-MARC CHOMAZ, LadHyX, Ecole Polytechnique — We investigate numerically the transient linear growth of three-dimensional perturbations in homogeneous time-evolving hyperbolic tangent mixing layers. We identify perturbations, which are optimal in terms of their kinetic energy gain, over a range of finite, predetermined time intervals. We consider a time-dependent two-dimensional base flow associated with the growth and nonlinear saturation of two wavelengths of the classical "Kelvin-Helmholtz instability" (KHI), allowing for the eventual merger of two elliptical KHI billows into a larger single elliptical vortex. If the time-evolving flow actually involves substantial evolution of the primary KHI during the optimization time interval, two broad classes of inherently 3D linear optimal perturbations arise, associated at low wavenumbers with the well-known core-centred elliptical translative instability, and at higher wavenumbers with the braid-centred hyperbolic instability. The growth of the elliptical secondary perturbations is strongly suppressed during primary KHI merger, due to the significant disruption of the primary billow cores, while hyperbolic perturbations, localized in the braid region between the two merging KHI billows, can still undergo significant transient energy growth.

9:18AM A10.00007 Minimal seeds in mixing layers , SAMUEL RABIN, DAMTP, University of Cambridge, COLM CAULFIELD, BP Institute & DAMTP, University of Cambridge, RICHARD KERSWELL, School of Mathematics, University of Bristol — Recent studies on transition have investigated the nonlinear transient growth properties of the Navier-Stokes equations by using variational techniques to optimize perturbation structure in order to reveal the "minimal seed" for turbulence (Cherubini et al 2010, Monokrousos et al 2011, Pringle et al 2012). These studies were performed on geometries and Reynolds numbers that were linearly stable, yet experimental results demonstrated could transition to turbulence, such as Plane Couette Flow (PCF). In contrast to PCF, one of the most commonly studied transition mechanisms is the Kelvin-Helmholtz instability (KHI) of inflectional shear layers, which layers can be shown to be unstable to infinitesimal perturbations for quite moderate Reynolds numbers. In this study, we apply the recently developed variational techniques to optimize the kinetic energy over finite time horizons of three-dimensional finite amplitude perturbations for a time-evolving background flow initially described by a hyperbolic tangent function, which flow is subject to KHI. Our objective is to determine what are the minimal energy perturbations which can trigger turbulence in this geometry, and what role is played by KHI in such "optimized" transition. This research was supported by EPSRC.

9:31AM A10.00008 The mixing layer downstream of a "Λ"-notched splitter plate , LUTZ TAUBERT, EMILE SUEHIRO, ISRAEL WYGNANSKI, The University of Arizona — The turbulent mixing layer created downstream of a "Λ"-notched splitter plate that was aligned with the free stream and whose trailing edge was inclined at 60° to the flow was investigated experimentally at two velocity ratios. It was observed that the rate of spread of this mixing layer relative to its local center was identical to the rate of spread of a two dimensional mixing layer provided all distances were measured from the trailing edge. Harmonic excitation was applied to this base flow by means of flaperons mounted on the trailing edges of the splitter plate. The external excitation enabled the separation of the instability wave fronts originating from the two opposing trailing edges of the "Λ"-notch. The effects of excitation frequency, amplitude and phase between the oscillating flaperons on the spreading rate and the orientation and velocity of the large coherent structures in the mixing layer were determined and the variation of the wave front angles was analyzed along their paths.

Sunday, November 18, 2012 8:00AM - 9:44AM –

Session A11 Bubbles I 26A - Chair: Francisco Pereira, CNR-INSEAN, The Italian Ship Model Basin

8:00AM A11.00001 Formation of Micro-Scale Gas Pockets From Underwater Wall Orifices[1] , FRANCISCO A. PEREIRA, CNR-INSEAN, The Italian Ship Model Basin, MORTEZA GHARIB, California Institute of Technology — Our experiments examine the formation of micro-scale gas pockets from orifices on walls with hydrophilic and hydrophobic wetting properties. Bubble injection is operated in a liquid at rest at constant flow rate and in a quasi-static regime, and the mechanism of bubble growth is investigated through high speed recordings. The growth dynamics is studied in terms of orifice size, surface wetting properties and buoyancy sign. The bubble formation is characterized by an explosive growth, with a pressure wave that causes the bubble to take highly transient shapes in its very initial stages, before stabilizing as a sphere and growing at a relatively slow rate. In case of positive buoyancy, the bubble elongates with the formation of a neck before detaching from the wall. When buoyancy acts towards the wall, the bubble attaches to the wall and expands laterally with a moving contact line. In presence of hydrophobic surfaces, the bubble attaches immediately to the wall irrespective of buoyancy direction and takes a hemispherical shape, expanding radially along the surface. A force balance is outlined to explain the different figures.

[1]The work was performed by FAP while on leave from CNR-INSEAN, and is supported by the Office of Naval Research (ONR).

8:13AM A11.00002 Writing bubbles , SANDER WILDEMAN, HENRI LHUISSIER, CHAO SUN, ANDREA PROSPERETTI, DETLEF LOHSE, University of Twente — We report on the nucleation of bubbles under a solid sphere immersed in a supersaturated liquid that is gently rubbed against a surface. For a fixed liquid supersaturation, bubbles are observed only above a certain rubbing velocity threshold. Above this threshold and provided that bubbles adhere better to the surface than to the sphere, a regularly spaced row of growing bubbles is left behind on the surface. Direct observation through a transparent sphere shows that each bubble in the row actually results from the early coalescence of several microscopic bubbles, which nucleate between the sphere and the surface. Together with the influence of the degree of supersaturation and the normal force between sphere and surface, we study the influence of the liquid itself (water or ethanol), the sphere material (glass, metal or Teflon) and of the surface roughness (polished or unpolished). Regardless of its precise origin, this method of "writing bubbles" also provides a simple way to spatially and temporally control the nucleation of bubbles on a surface and to study their interactions.

8:26AM A11.00003 The Influence of Injection Angle on Bubble Formation from a Micro-Pillar[1] , FARZAD HOUSHMAND, DAREN ELCOCK, YOAV PELES, Rensselaer Polytechnic Institute — Bubble formation in a microchannel in the presence of a 150 μm diameter micro-pillar was investigated. Nitrogen stream was injected into water flow in a 225 μm deep, 1500 μm wide, and 27.5 mm long horizontal microchannel through 20 μm slits cut on the micropillar located vertically in the centerline of the channel. Bubble formation in different devices with varying slit angles—with respect to liquid flow—of 0°, $\pm$30°, $\pm$80°, $\pm$110°, and 180° were studied for liquid flow rates of 13, 34 and 54 ml/min, and gas flow rates ranging from 0.5 to 7 ml/min. Based on high speed high magnification imaging, three distinct formation modes were observed depending on the slit angle and liquid and gas flow rates: discrete bubbling, attached ligament, and mixed modes. Micro-PIV technique was used to study the liquid flow in vicinity of the pillar to elucidate the phenomena controlling bubble formation.

[1]This work is supported by the Office of Naval Research (Program Manager Dr. Mark Spector).

8:39AM A11.00004 Highly time-resolved measurement for bubble nucleation induced by femtosecond laser pulses , YUKI MIZUSHIMA, Graduate School of Engineering, Shizuoka University, TAKAYUKI SAITO, Graduate School of Science and Technology, Shizuoka University — Femtosecond laser pulses (fs pulses) lead very interesting phenomena due to their extremely high energy density. The effects on substances are not thermal, but are multi-photon absorption. When this multi-photon absorption of fs pulses operates on water, extraordinary phenomena different from laser-induced cavitation by a usual laser such as nano-pulse laser are induced. In this study, fs pulses were focused on ultra-purified water in a glass cell through several types of lens. Some fs pulses split from original beams through a beam splitter were used as probing light of femtosecond order. Femtosecond-order time-resolved optical measurement was realized by adjusting a light path length of the probing light. We found out strange time-series process of refraction index changes of the water irradiated by the fs pulses, and the bubble nucleation and bubble growth, and the interesting bubble properties. Based on these results, we discuss a relation between those and fs-pulse peak intensity. Further, we discuss the nucleation and growth process from femtoseconds to picoseconds.

8:52AM A11.00005 Simulated Collapse of Small Bubble Clusters on a Wall , ARPIT TIWARI, CARLOS PANTANO, JONATHAN B. FREUND, University of Illinois at Urbana-Champaign — The dynamics of bubble clusters have largely been studied either via simplified continuum models for bubbly liquids that augment single-phase system with equations for spherical bubble dynamics (forgoing shape distortions and jetting) or by assuming the liquid to be incompressible. However, these simplifications are not fully applicable, even for a near-wall single-bubble collapse. It is expected that jetting and acoustic emissions potentially play an important role, for example, in damage to the wall. We present simulations in which up to 50 bubbles interact in a compressible liquid adjacent a wall. A new numerical scheme enables these simulations by using a diffuse-interface-based Eulerian approach with a multiphase mixture model that maintains interface integrity without requiring the geometric description demanded in typical interface tracking methods. The properties of this scheme and its integration in an adaptive mesh refinement framework facilitate simulation of three-dimensional clusters. We will discuss the cluster dynamics, including the collapse and re-expansion history, and particularly assess pressure on the wall as potential surrogate for damage. We see strong, though at times significantly asymmetric and transitory, focused high and low pressures on the wall.

9:05AM A11.00006 Simulations of surfactant effects on the coalescence of drops and bubbles[1] , DAVID MARTIN, FRANCOIS BLANCHETTE, University of California Merced — We present simulations of coalescence in the presence of surfactant. We assume axial symmetry, and consider a fluid-fluid interface on which surfactant concentration and mass are tracked as functions of arclength. Our model can account for two physically distinct setups: a soap bubble merging with a suspended soap film; and a surfactant covered liquid drop merging with a reservoir. In both cases, we describe the regime in which coalescence is only partial. Along with viscous effects, represented by the Ohnesorge number, the elasticity of the surface tension relative to the surfactant concentration is seen to play a key role, and exhibits a surprising nonmonotonic influence. Effects of gravity are also simulated, along with effects of differing initial conditions, including uneven initial surfactant concentration, as is likely to arise in physical applications.

[1]We acknowledge support from NSF grant DMS 0808129

9:18AM A11.00007 Large Bubble Rupture Sparks Fast Liquid Jet , THOMAS SEON, ARNAUD ANTKOWIAK, Institut d'Alembert, UPMC & CNRS, Jussieu, Paris — The novel experimental observation of long and narrow jets shooting out in disconnecting large elongated bubbles is presented. We investigate this phenomenon by carrying out experiments varying the control parameters and we propose a universal scaling law for the jet velocity, which unexpectedly involves the bubble height to the power 3/2. This anomalous exponent suggests an energy focusing phenomenon. We demonstrate experimentally that this focusing is purely gravity-driven and independent of the pinch-off singularity.

9:31AM A11.00008 Coalescence cascade of bubbles , FENGHUA ZHANG, King Abdullah University of Science and Technology, Thuwal, Saudi Arabia, PETER TABOREK, Department of Physics and Astronomy, University of California, Irvine, California, USA, SIGURDUR THORODDSEN, King Abdullah University of Science and Technology, Thuwal, Saudi Arabia — Coalescence of two bubbles produces satellite bubbles, known as partial coalescence. The coalescence of the satellite with the main bubble may produce even smaller satellites, so that the coalescence proceeds like a cascade. While the coalescence cascade of a drop has been well known, we show here for the first time that bubbles can do so too. By performing experiments using high-speed imaging and the non-dimensional analysis, the sizes and formation time of the generated satellites are characterized and explained. In addition, we have observed satellites formation at focusing of multiple waves for the first time, as well as formation of several sub-satellites. The conditions of their formation are identified. These findings are important for multi-phase fluid flows, like foams or emulsions, separation of immiscible fluids, printing and painting, etc.

Sunday, November 18, 2012 8:00AM - 9:44AM –

Session A12 Vortex I 26B - Chair: Arindam Banerjee, Lehigh University

8:00AM A12.00001 Accuracy of current actuator-line modeling methods in predicting blade loads and wakes of wind turbines , PANKAJ JHA, Pennsylvania State University, MATTHEW CHURCHFIELD, PATRICK MORIARTY, National Renewable Energy Laboratory, SVEN SCHMITZ, Pennsylvania State University — Actuator-line modeling has become a prominent method for computing individual wind turbine wakes and their interaction within large wind turbine arrays. The advantage of actuator-line modeling is rooted in discretizing wind turbine blades as compact lines of body forces within an outer RANS- or LES-type solver. This eliminates the need for prohibitively expensive fully-resolved and detailed blade flow simulations. However, the close linkage between sectional blade forces and their reactive momentum deficit distribution just behind the rotor and its evolution and recovery further downstream in the wake has not been addressed in much detail in the literature. We have observed that current actuator-line modeling overpredicts blade tip loads by as much as twenty percent in comparison to blade-element momentum analyses and available data. The effect of the observed discrepancies in blade loading on the wake recovery and blade loading of a downstream wind turbine is unknown. This work addresses current efforts in quantifying the uncertainty and improving the predictive capability of actuator-line modeling. A detailed study of the combined effects of local grid refinement, body force projection width, and actuator element distribution is presented.

8:13AM A12.00002 Vertical Axis Wind Turbine flows using a Vortex Particle-Mesh method: from near to very far wakes[1] , STEPHANE BACKAERT[2], PHILIPPE CHATELAIN, GREGOIRE WINCKELMANS, Universite catholique de Louvain (UCL), Institute of Mechanics, Materials and Civil Engineering (iMMC), STEFAN KERN, THIERRY MAEDER, DOMINIC VON TERZI, GE Global Research, WIM VAN REES, PETROS KOUMOUTSAKOS, ETH Zurich, CSElab — A Vortex Particle-Mesh (VPM) method with immersed lifting lines has been developed and validated. The vorticity-velocity formulation of the NS equations is treated in a hybrid way: particles handle advection while the mesh is used to evaluate the differential operators and for the fast Poisson solvers (here a Fourier-based solver which simultaneously allows for unbounded directions and inlet/outlet boundaries). Both discretizations communicate through high order interpolation. The immersed lifting lines handle the creation of vorticity from the blade elements and its early development. LES of Vertical Axis Wind Turbine (VAWT) flows are performed, with a relatively fine resolution (128 and 160 grid points per blade) and for computational domains extending up to $6\,D$ and $14\,D$ downstream of the rotor. The wake complex development is captured in details, from the blades to the near wake coherent vortices, to the transitional ones, to the fully developed turbulent far wake. Mean flow statistics in planes (horizontal, vertical and cross) are also presented. A case with a realistic turbulent wind inflow is also considered. The physics are more complex than for HAWT flows.

[1]Computational resources provided by a PRACE award

[2]Research Assistant (PhD student)

8:26AM A12.00003 Three dimensional visualization of the interaction between energetic coherent motions and tip vortices in the wake of an axial-flow marine turbine , DANIEL TROOLIN, Fluid Mechanics Research Instruments, TSI Incorporated, Shoreview, MN, USA, LEONARDO CHAMORRO, SEUNG-JAE LEE, ROGER ARNDT, FOTIS SOTIROPOULOS, St. Anthony Falls Laboratory, College of Science and Engineering, University of Minnesota — Tip vortices of axial-flow turbines play a key role in modulating the mean and turbulent characteristics of the wake. Understanding their evolution and the mechanisms that trigger instability is crucial to improve bulk power extracted in a wind farm. In this study, we study the interaction between the tip vortices generated by a miniature axial-flow turbine and strong coherent motions present in the flow. The turbine was placed in a water flume at the St. Anthony Falls Laboratory at the University of Minnesota under subcritical conditions. A circular cylinder was placed upstream of the turbine to induce the energetic coherent structures in the flow. Three-dimensional three-component (3D3C) velocity measurements were made in the flow downstream of the miniature turbine. The focus was placed on visualizing the complex interaction between the von Karman vortices shed by the cylinders and the turbine tip vortices. New insights on the tip vortex dynamics and three dimensional characteristics of the wake flow will be discussed.

8:39AM A12.00004 Flow Behavior Around Coupled, Rotating Turbines in Steady Flow[1] , MATTHEW FU, JOHN DABIRI, Caltech — Counter-rotating vertical axis turbines (VATs) have been shown to yield increased power density in wind farms as compared to typical horizontal axis wind turbine (HAWT) farms. However, the governing physical mechanisms remain poorly understood. Scale model experiments in a free-surface water tunnel were conducted to characterize the effect of parameters such as turbine separation, tip speed ratio, and flow speed on the downstream flow field and the resulting vortex shedding from VATs. The flow field was visualized using particle image velocimetry (PIV) and planar laser induced fluorescence. The results are compared and contrasted with recent studies of counter-rotating circular cylinders to determine if suppression of vortex shedding plays a similarly important role in dictating the overall wake dynamics.

[1]This research was made possible through the generosity of Gordon and Betty Moore Foundation and the Caltech SURF Program

8:52AM A12.00005 Breaking the Symmetry with Flexible Blades[1] , JULIA COSSE, DAEGYOUM KIM, MORTEZA GHARIB, California Institute of Technology — Savonious turbines take advantage of shapes that experience higher drag when moving with the wind and lower drag when moving against the wind. Generally curved blades (e.g. semi circles) have supplied this characteristic drag differential, which allows the turbine to spin. While flexibility is often associated with drag reduction via reconfiguration, it is less well known that this same mechanism can be used for drag enhancement. Inspired by these unique properties we designed a turbine and placed it in a wind tunnel to further investigate the potential of flexible materials. The model was built such that it can be equipped with blades made out of different materials and the blade pitch angle can be chosen arbitrarily. As expected, the turbine didn't rotate when rigid blades were fixed to the turbine because both sides of the turbine experienced identical drag forces. However, when flexible blades were used, the wind reconfigured the shape of the blades such that there was a drag differential across the turbine which resulted in rotation. The characteristics of the flexible blades will be described in detail along with an analysis of their performance.

[1]This research is supported by the Gordon and Betty Moore foundation.

9:05AM A12.00006 Optimum design of Hydrokinetic turbine based on Fluid structure interaction analysis[1] , NITIN KOLEKAR, ARINDAM BANERJEE, Lehigh University, Bethlehem, PA — Hydrokinetic turbines, unlike conventional hydraulic turbines are zero head energy conversion devices, which utilize the kinetic energy of flowing water for power generation. Though the basic operation is similar to wind turbines, due to denser working media, these turbines are subjected to higher loads and stresses. The present work aims at hydrodynamic design and coupled fluid structure interaction (FSI) analysis for a horizontal axis hydrokinetic turbine. Blade element momentum (BEM) theory is utilized to analyze fluid forces and torque developed on turbine blades. The results of BEM are compared with a detailed three-dimensional computational fluid dynamics (CFD) analysis. The CFD domain is coupled with the structural domain using arbitrary Lagrangian-Eulerian scheme to find stresses in turbine components. A parametric study will be carried out to understand the effect of various parameters like blade pitch angle, flow velocity and RPM on the stresses developed on blades for different blade materials (aluminum and steel). Based on the one-way FSI analysis, the flow conditions and turbine design parameters will be optimized to achieve maximum possible efficiency.

[1]Authors acknowledge financial support through ONR Grant # N000141010923.

9:18AM A12.00007 The Influence of Spanwise Flow on Leading-Edge Vortex Growth , JAIME WONG, JOCHEN KRIEGSEIS, DAVID RIVAL, University of Calgary — It has been postulated that a spanwise component velocity through the core of a leading-edge vortex (LEV) can limit its growth and allow the LEV to remain attached to the wing. In the case of a delta wing, spanwise velocity is produced by wing sweep. However, in the case of flapping-wing flight, centripetal and Coriolis accelerations produce spanwise velocities which vary periodically. In order to understand the effect of various spanwise velocity profiles on LEV growth a simple analytical model for vortex growth has been developed. This model is based on the transport of vorticity-containing mass into the LEV through the leading-edge shear layer. By first neglecting spanwise effects, the model has been verified against a nominally two-dimensional plunging profile using Particle Image Velocimetry (PIV). With the addition of a spanwise transport of vorticity-containing mass, swept and flapping spanwise velocity profiles have been modeled and compared with three-dimensional, three-component velocity data collected using Particle Tracking Velocimetry.

9:31AM A12.00008 Global Vorticity Shedding on Rectangular and Streamlined Foil Geometries[1] , STEPHANIE STEELE, Massachusetts Institute of Technology, JASON DAHL, University of Rhode Island, GABRIEL WEYMOUTH, Singapore MIT Alliance for Research and Technology, MICHAEL TRIANTAFYLLOU, Massachusetts Institute of Technology — We explore several aspects of the fluid phenomenon we call global vorticity shedding. Global vorticity shedding occurs when an object in a fluid with circulation suddenly vanishes, shedding the entirety of the boundary layer vorticity into the wake at once. Global vorticity shedding is in distinct contrast with traditional massive separation shedding, in which vorticity is shed from a body from only a few separation points into the fluid. In our experiments, we approximate the disappearance of a towed foil by rapidly retracting the foil in the span-wise direction. We show that for a square-tipped vanishing foil at an angle of attack, the globally shed boundary layer vorticity forms into primary vortices, which evolve and eventually amalgamate with secondary vortices to leave two lasting vortices in the wake. The secondary vortices are a result of three-dimensionality in the flow. We further explore streamlined foil geometries to achieve a simpler and less three-dimensional wake. Vortex formation times are small, with vortices fully formed nearly instantaneously in the flow, making the application of global vorticity shedding promising for a force transducer to impart large and fast maneuvering forces on an underwater vehicle.

[1]This research was made with Government support under and awarded by DoD, Air Force Office of Scientific Research, National Defense Science and Engineering Graduate (NDSEG) Fellowship, 32 CFR 168a.

Sunday, November 18, 2012 8:00AM - 9:44AM –

Session A13 Geophysical: General I 27A - Chair: Paul Linden, University of Cambridge

8:00AM A13.00001 Numerical Simulation of the tidal effects on estuarine circulation in the San Juan Bay , EDGARDO GARCIA, MIGUEL CANALS, University of Puerto Rico, Mayaguez Campus, JORGE CAPELLA, JULIO MORELL, Caribbean Coastal Ocean Observing System, STEFANO LEONARDI, University of Puerto Rico, Mayaguez Campus, CARIBBEAN COASTAL OCEAN OBSERVING SYSTEM COLLABORATION — The regional oceanic modeling system ROMS has been implemented in San Juan Bay, Puerto Rico, to investigate quantitatively the mixing processes and as a forecast tool to support emergency planning and resource management in the area. The response of the San Juan Bay circulation to both river discharges and tidal forcing has been investigated. A hind-cast simulation is performed and compared with time series measurements and hidrographic data to validate the model. Sensitivity studies to turbulence mixing parameterization have been carried out under different forcing scenarios. A simulation without river outflow but forced with tidal constituents along the open boundaries is performed. Good agreement has been found with coastal observations with amplitudes gauges and modeled amplitudes constituents. A numerical experiment of the response of the Bay's circulation to river discharge only is performed and compared with the tide plus river forcing and tidal forcing only scenarios. Salinity distributions and vertical mixing are affected by the proximity to deep waters of the San Juan Bay Estuary, also the bulge region and plume structure in the entrance of San Juan Bay are highly affected by the bottom friction for the case with tide forcing.

8:13AM A13.00002 Cross-shore thermally-driven exchange on two coral reef shorelines[1] , GENO PAWLAK, University of California, San Diego, LAUREN TUTHILL, WELLS JUDITH, MARK MERRIFIELD, University of Hawaii at Manoa, STEPHEN MONISMITH, Stanford University — The dynamics of cross-shelf circulation influence the distribution of heat, salt, nutrients, contaminants, sediment, and planktonic organisms in the nearshore coastal ocean. For tropical reef coastlines, the horizontal redistribution of heat by cross-shelf circulation moderates the daily variations in temperature experienced by coral polyps, potentially reducing thermal stress. A key mechanism for this exchange is thermally driven baroclinic circulation. Here, observations from two reefs, at Eilat, Israel and Oahu, Hawaii, are presented that highlight the role of thermally forced exchange in cross-shore transport for distinct dynamic regimes. At each site, daytime conditions are characterized by offshore flow at the surface in response to increased temperatures in shallower water near shore. Nighttime cooling results in offshore flow near the bed. While an advection-dominated thermal balance provides a good description of the flows observed off Eilat by Monismith et al. (2006), phase differences between the heat flux, the thermal response and the cross-shore flows indicate that the Oahu flow is in an unsteady regime. Estimates of momentum and thermal balance terms from field data further confirm differences in flow regimes.

[1]National Science Foundation

8:26AM A13.00003 Wave, Current and Bottom Topographical Interactions in the Coastal Ocean Bottom Boundary Layer[1] , ADITYA NAYAK, CHENG LI, DANIEL CHOI, JOSEPH KATZ, Johns Hopkins University — PIV measurements were performed in the inner part of the coastal ocean bottom boundary layer (BBL), at a depth of 20m. 2D velocity distributions with resolution of 4.5 mm were obtained in two 28×28 cm^2 planes, the first aligned with the current, and the second with the dominant wave direction. Filtering the reflection from the bottom facilitated velocity measurements starting from 3 mm above the seabed, fully resolving the inner part of the wave and current boundary layers. Co-located acoustic doppler velocimeter measurements were used to calculate Reynolds stress profiles by filtering out wave-induced motions from the PIV data. High-resolution sonar was used to map the bottom topography, and characterize the roughness. Several datasets, some spanning an entire tidal cycle were obtained at 6 Hz, under varying relative wave-current magnitude and directions, as well as ripple orientations. The PIV data resolved the interaction of currents and waves with the roughness, and suggested, consistent with the Grant and Madsen model, that the BBL contained two log layers with different slopes. The thicker and milder sloped log layer was part of the mean current boundary layer. Below it, a thinner layer with a higher slope was part of the wave boundary layer.

[1]Acknowledgment: NSF

8:39AM A13.00004 Applications the Lagrangian description in aperiodic flows[1] , CAROLINA MENDOZA, Universidad Politecnica de Madrid, ANA MARIA MANCHO, ICMAT (CSIC-UAM-UC3M-UCM) — We use several recently developed Lagrangian tools for describing transport in general aperiodic flows. In our approach the first step is based in a Lagrangian descriptor (the so called function M). It measures the length of particle trajectories on the ocean surface over a given interval of time. We describe its output over satellite altimetry data on the Kuroshio current. The technique is combined with the direct computation of manifolds of Distinguished Hyperbolic trajectories and a very detailed description of transport is achieved across an eddy and a jet on the Kuroshio current [1,2]. A second velocity data set is examined with the M function tool. These are obtained from the HYCOM project on the Gulf of Mexico during the time of the oil-spill. We have identified underlying Lagrangian structures and dynamics. We acknowledge to the hospitality of the university of Delaware and the assistance of Bruce Lipphardt and Helga Huntley in accessing the model data sets.

[1] C. Mendoza, A.M. Mancho. Physical Review Letters 105 (2010), 3, 038501-1-038501-4.
[2] C. Mendoza, A.M. Mancho. The Lagrangian description of aperiodic flows: a case study of the Kuroshio Current. acepted in Nonlinear Proc. Geoph

[1]We acknowledge to the grants: UPM-AL12-PAC-09, Becas de Movilidad de Caja Madrid 2011, MTM2011-26696 and ILINK-0145.

8:52AM A13.00005 Formation and fate of contaminant particles controlled by turbulent coherent structures and geochemistry in a reactive river confluence[1] , CRISTIAN ESCAURIAZA, CHRISTIAN GONZALEZ, PAULA GUERRA, PABLO PASTEN, GONZALO PIZARRO, Pontificia Universidad Catolica de Chile — A river confluence in a 40-degree angle and with a high momentum ratio (M=12.8) generates the hydrodynamic mechanisms that control the formation and fate of arsenic-rich particles in an extremely arid region in northern Chile. Based on the conditions measured in the field, we carry out detached-eddy simulations (DES) and laboratory experiments for a simplified confluence, providing new insights on the effects of the streamwise helical vortices and recirculating regions on the turbulent mixing. A Lagrangian model is also developed to study the influence of large-scale coherent structures on particle transport, computing statistics of trajectories in the flow field and determining the characteristic time-scales of the flow. This investigation helps to clarify the complex interactions between the 3D vortical structures in the flow field and the geochemical reactions, which are the controlling mechanisms of particle formation and fate of the contaminants in the river.

[1]Supported by Fondecyt Project 1100943 and ONR-G NICOP Project N622909-11-1- 7041.

9:05AM A13.00006 Water-wave diffraction by small undulation on a porous ocean-bed in the presence of surface tension in a two-layer fluid , SUBASH CHANDRA MARTHA, SRIKUMAR PANDA, Indian Institute of Technology Ropar, India — The problem involving wave scattering in a two-layer fluid due to small bottom undulation on the porous ocean-bed is investigated within the framework of two-dimensional linearized water wave theory in the presence of surface tension at the upper free surface. In each layer it is assumed that the flow is irrotational and the fluid is inviscid and incompressible. In such a two-layer fluid there exist waves with two different modes, one with lower wave number propagate in the upper layer whilst those with higher wave number propagate in the lower layer. An incident wave of a particular mode gets reflected and transmitted by the undulating bottom into waves of both modes. By employing perturbation analysis in conjunction with Fourier transform method, the governing BVPs are solved and the reflection and transmission coefficients are obtained in terms of integrals involving the shape function $c(x)$ representing the bottom undulation. These coefficients can be evaluated once the shape function is known. One special type of undulating bottom is considered as an example to evaluate the related coefficients in detail. These coefficients are depicted graphically to demonstrate the transformation of energy between the two different modes and also to validate the theoretical observations.

9:18AM A13.00007 Gravity currents in strongly stratified fluids , BENJAMIN MAURER, PAUL LINDEN, DAMTP, University of Cambridge — For fluids subject to gravity, vertical density gradients are stable while horizontal density gradients often sharpen to laterally propagating fronts called gravity currents. As recent studies have shown, the energetics and therefore the dynamics of gravity currents propagating in a stratified fluid are affected by the vertical density stratification. To investigate the role of vertical stratification in the flow dynamics, we conducted an experimental and numerical study of buoyancy-driven, high *Re* lock-releases between two fluids of equal density stratification structure and strength but differing depth-averaged densities. We examined both layered and continuous vertical density stratifications, focusing on the cases where the vertical density gradients are strong relative to the horizontal mean density difference. Lock-releases between discretely layered fluids of unequal depth-averaged densities result in multiple interleaving flows advected by a less energetic full-depth flow. Lock-releases between linearly stratified fluids of unequal depth-averaged densities result in a full-depth flow similar to the less energetic flow in the discretely layered case. We present dimensional analysis and energy scaling models describing the relevant length scales and buoyancy forcing

9:31AM A13.00008 A radar backscattering mechanism of ocean surface in response to rainfall[1] , XINAN LIU, QUANAN ZHENG, REN LIU, JAMES H. DUNCAN, University of Maryland, College Park — The characteristics of ocean surface in response to rainfall and its radar back-scatter are simultaneously measured in 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, secondary droplets and ring waves are measured with a cinematic Laser-Induced-Florescence (LIF) technique. Our experimental results show that impinging raindrops on the water surface generate various water surface structures with different relative sizes. Among them stalks and crowns comprise the dominant radar backscattering. On the basis of these laboratory experiments and theories of radar scattering from a rough surface, a near-resonance radar backscattering model for quantifying the dependence of the radar return intensity on rain rate on the ocean surface is developed. The model explains the radar response to rain rate simultaneously observed by C-band ASAR and ground-based weather radar. The physical model provides reasonable mechanisms to explain the frequency dependence and polarization behavior of radar signatures from rain cells on the ocean surface.

[1]This work is supported by the National Science Foundation, Division of Ocean Sciences under grant OCE962107.

Sunday, November 18, 2012 8:00AM - 9:44AM –

Session A14 General Experiments I 27B - Chair: Geoffrey Spedding, University of Southern California

8:00AM A14.00001 Multi-variable calibration of temperature estimation in individual non-encapsulated thermo liquid crystal micro particles[1] , RODRIGO SEGURA, CHRISTIAN CIERPKA, MASSIMILIANO ROSSI, CHRISTIAN J. KÄHLER, Institute for Fluid Mechanics and Aerodynamics Bundeswehr University Munich — An experimental method to track the temperature of individual non-encapsulated thermo-liquid crystal (TLC) particles is presented. TLC thermography has been investigated for several years but the low quality of individual TLC particles, as well as the methods used to relate their color to temperature, has prevented the development of a reliable approach to track their temperature individually. In order to overcome these challenges, a Shirasu Porous Glass (SPG) membrane approach was used to produce an emulsion of stable non-encapsulated TLC micro particles, with a narrower size distribution than that of encapsulated TLC solutions which are commercially available (Segura et al, Microfluid Nanofluid, 2012). On the other hand, a multi-variable calibration approach was used, as opposed to the well known temperature-hue relationship, using the three-components of the HSI color space measured in each particle image. A third degree three-dimensional polynomial was fitted to the color data of thousands of particles to estimate their temperature individually. The method is able to measure individual temperatures over a range exceeding the nominal range of the TLC material, with lower uncertainty than any method used for individual particle thermography reported in the literature.

[1]Financial support from the German Research Foundation (DFG), under the Forschergruppe 856 grant program, is gratefully appreciated.

8:13AM A14.00002 Laboratory investigation of the erosion of cohesive sediments under oscillatory flows using a synchronized imaging technique , IN MEI SOU, JOSEPH CALANTONI, ALLEN REED, YOKO FURUKAWA, Naval Research Laboratory — A synchronized dual stereo particle image velocimetry (PIV) measurement technique is used to examine the erosion process of a cohesive sediment core in the Small Oscillatory Flow Tunnel (S-OFT) in the Sediment Dynamics Laboratory at the Naval Research Laboratory, Stennis Space Center, MS. The dual stereo PIV windows were positioned on either side of a sediment core inserted along the centerline of the S-OFT allowing for a total measurement window of about 20 cm long by 10 cm high with sub-millimeter spacing on resolved velocity vectors. The period of oscillation ranged from 2.86 to 6.12 seconds with constant semi-excursion amplitude in the test section of 9 cm. During the erosion process, Kelvin–Helmholtz instabilities were observed as the flow accelerated in each direction and eventually were broken down when the flow reversed. The relative concentration of suspended sediments under different flow conditions was estimated using the intensity of light scattered from the sediment particles in suspension. By subtracting the initial light scattered from the core, the residual light intensity was assumed to be scattered from suspended sediments eroded from the core. Results from two different sediment core samples of mud and sand mixtures will be presented.

8:26AM A14.00003 A new method to determine the yield stress of diluted polymeric solutions , ENRIQUE SOTO, SERVANDO RUIZ, MARIA SOLEDAD CORDOVA AGUILAR, Universidad Nacional Autonoma de Mexico — A new method to measure the yield stress for diluted polymeric solutions is presented. The tested solutions exhibit shear thinning behavior a once the critical yield stress is overcame. In rheology, these fluids are known as Herschel-Buckley. The yield stress phenomenon and its relation with bubble motion is an important issue for different industries, for example, personal care, paints and some others. As a result of the yield stress, small bubbles remain trapped in the fluid bulk, but above a critical volume, which is related with the characteristic yield stress, the bubbles flow in the liquid. In order to change the bubble volume, the liquid is placed in a cylindrical container whose pressure is decreased by a vacuum pump. The bubble growths as the pressure decreases and keeps its position until it reaches the critical volume. The bubble shape changes with volume and velocity, and a competition among surface, gravitational, inertial and viscous forces is discussed. The yield stress determined value is higher than the obtained from simple shear measurements due to the complex flow around the bubble.

8:39AM A14.00004 Plasma Adaptive Optics Characterization using Dispersive FTIR Interferometry , BRIAN NEISWANDER, ERIC MATLIS, THOMAS CORKE, University of Notre Dame — Previous work by the authors has investigated the implementation of plasma as an adaptive optic element. Plasma has no moving parts, high frequency response, and an index of refraction that is dependent on the applied voltage potential. The refractive index is a function of the electron density and heavy particle density, and the accurate measurement of these values is critical to the development of plasma optical devices. Traditionally, such measurements are achieved by probing plasma with multiple wavelengths or by imposing specific assumptions on the heavy particle density. This work uses dispersive Fourier transform infrared (FTIR) interferometry to probe a plasma optic element and measure its optical properties across a wide range of wavelengths. This approach is based on a least-squares reconstruction of the plasma electron density and heavy particle density values. Both theoretical modeling and experimental validation are presented.

8:52AM A14.00005 Visualization of Capsule Reentry Vehicle Heat Shield Ablation using Naphthalene Planar Laser-Induced Fluorescence Imaging[1] , CHRISTOPHER COMBS, NOEL CLEMENS, The University of Texas at Austin, PAUL DANEHY, NASA Langley Research Center — NASA has continued interest in the study of ablation owing to the need to develop suitable thermal protection systems for spacecraft that undergo planetary entry. Ablation is a complex multi-physics process, and codes that predict it require a number of coupled submodels, each of which requires validation. For example, Reynolds-averaged Navier Stokes (RANS) and large-eddy simulation (LES) codes require models of the turbulent transport of ablation products under variable compressibility and pressure gradient conditions. A new technique has been developed at The University of Texas at Austin that uses planar laser-induced fluorescence (PLIF) of a low-temperature sublimating ablator (naphthalene) to enable visualization of the ablation products as they are transported in a boundary layer. While high temperature ablation is extremely difficult to recreate in a laboratory environment, low temperature ablation creates a limited physics problem that can be used to simulate the ablation process. In the current work a subscale capsule reentry vehicle model with a solid naphthalene heat shield is tested in a Mach 5 wind tunnel. PLIF imaging reveals the distribution of the ablation products as they are transported into the boundary layer and over the capsule shoulders.

[1]Work supported by NASA Space Technology Research Fellowship Program under grant NNX11AN55H.

9:05AM A14.00006 Development of a Digital Fringe Projection Technique to Quantify the Transient Behavior of Wind-Driven Surface Droplet/Rivulet Flows[1] , KAI ZHANG, SONG ZHANG, HUI HU, Iowa State University — A novel digital fringe projection (DFP) technique is developed to achieve non-intrusive thickness measurements of wind-driven water droplet/rivulet flows. The DFP 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. The digital projector is used to project a fringe pattern of known characteristics onto a test object (i.e., the water droplet/rivulet on the test plate). Due to the 3D shape profile of the test object, the fringe pattern is deformed seen from a perspective different from the projection axis. By comparing the distorted fringe pattern over the test object and a reference fringe pattern on a reference plane, the 3D profile of the test object with respect to the reference plane (i.e., the thickness distribution of the water droplet/rivulet flow) can be retrieved quantitatively and instantaneously. The DFP system is used to achieve time-resolved thickness distribution measurements of a droplet/rivulet flow driven by a boundary layer wind. The dynamic shape change and stumbling runback motion of the wind-driven water droplet/rivulet flow over the test plate are revealed clearly and quantitatively from the DFP measurement results. Such information is highly desirable to elucidate 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 NASA and NSF.

9:18AM A14.00007 Surface tension profiles in vertical soap films: an intrusive measurement , NICOLAS ADAMI, STÉPHANE DORBOLO, HERVÉ CAPS, Université de Liège-GRASP, GRASP TEAM — Soap films are known to be the basic constituent of complex fluid objects such as bubbles and foams. In order for their weight to be counterbalanced, those objects need to exhibit vertical surface tension profiles, which can only exist due to the presence of surfactant molecules at their interfaces. We here present a method that aims to probe vertical soap films surface tension profiles by use of elasto-capillary deformations. Both cases of free-draining and entertained soap films are investigated, leading the spatio-temporal behavior of the surface tension. We show that this behavior is highly dependent on the nature of the surfactant used to create soap films, emphasizing the influence of the physico-chemistry on the global behavior of the system. We propose order of magnitude calculations, which are in good agreement with experimental data.

9:31AM A14.00008 Measuring bubbles in a bubbly wake flow[1] , SEUNG-JAE LEE, ELLISON KAWAKAMI, ROGER E.A. ARNDT, Saint Anthony Falls Laboratory, University of Minnesota — This paper presents measurements of 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). A narrow depth-of-field (DoF) is required for imaging a 2-dimensional plane within a flow volume. Shadows of the bubbles were collected by a high-speed camera. Once a reference image, taken when no bubbles were present in the flow, was subtracted from the images, the image was segmented using an edge detection technique. The Canny algorithm was determined to be best suited for this application. A curvature profile method was employed to distinguish individual bubbles within a cluster of highly overlapping bubbles. The utilized algorithm was made to detect partly overlapping bubbles and reconstruct the missing parts. The movement of recognized individual bubbles was tracked on a two dimensional plane within a flow volume. In order to obtain quantitative results, the wake of a ventilated hydrofoil was investigated by applying the shadowgraphy technique and the described bubble detection algorithm. These experiments were carried out in the high speed cavitation tunnel at Saint Anthony Falls Laboratory (SAFL) of the University of Minnesota.

[1]This research is jointly sponsored by the Office of Naval Re- search, Dr. Ron Joslin, program manager, and the Department of Energy, Golden Field Office.

Sunday, November 18, 2012 8:00AM - 9:44AM –

Session A15 Biofluids: Micro-PIV 28A - Chair: Donald Webster, Georgia Institute of Technology

8:00AM A15.00001 Metachronal Propulsion, Hovering, and Signaling: High-Speed Tomographic PIV Measurements of Swimming Antarctic Krill , J. YEN, D.W. MURPHY, D.R. WEBSTER, Georgia Tech — Antarctic krill (*Euphausia superba*) are pelagic crustaceans that must swim continuously to avoid sinking. Krill swim by beating their five pairs of swimming legs (known as pleopods) in a metachronal pattern. Although metachrony is a common propulsion technique among crustaceans, the hydrodynamics of multiple appendages paddling in series has not been well investigated. Furthermore, the hydrodynamic signal created by the metachronally stroking pleopods is thought to play a role in schooling propensity among krill conspecifics. We present time-resolved tomographic PIV measurements of the flow generated by free-swimming Antarctic krill. Detailed flow measurements around the pleopods of a hovering Antarctic krill reveal flow being drawn backwards with each pleopod stroke in this drag-based swimming technique. Vortices forming around each pleopod pair during the power stroke also were found and may create additional thrust. Measurements in the wake of the krill reveal a pulsed jet flow with mean and oscillatory components. This wake signature may form a communication channel with nearby conspecifics and is discussed in the context of sensory ecology.

8:13AM A15.00002 Application of micro-PIV to the study of staphylococci bacteria bio-film dynamics , ERICA SHERMAN, University of Nebraska - Lincoln, KENNETH BAYLES, DEREK MOORMEIER, University of Nebraska - Medical Center, 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 staphylococcus bacteria without consequence, a localized staph infection has the potential to 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 grow in stable bio-films. Under other conditions, they organize in tower-like structures which break and are transported downstream by the flow. The fundamental questions addressed in this study 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 micro-PIV to study this problem. Bacteria were cultured in a glass microchannel and subjected to a range of steady shear rates. Micro-PIV measurements were made to map the flow over and around different types of bio-film structures. Measurements and control volume analysis will be presented quantifying forces acting on these structures.

8:26AM A15.00003 Roll and Yaw of Paramecium swimming in a viscous fluid , SUNGHWAN JUNG, SAIKAT JANA, Department of Engineering Science and Mechanics, Virginia Tech, MATT GIARRA, PAVLOS VLACHOS, Department of Mechanical Engineering, Virginia Tech — Many free-swimming microorganisms like ciliates, flagellates, and invertebrates exhibit helical trajectories. In particular, the Paramecium spirally swims along its anterior direction by the beating of cilia. Due to the oblique beating stroke of cilia, the Paramecium rotates along its long axis as it swims forward. Simultaneously, this long axis turns toward the oral groove side. Combined roll and yaw motions of Paramecium result in swimming along a spiral course. Using Particle Image Velocimetry, we measure and quantify the flow field and fluid stress around Paramecium. We will discuss how the non-uniform stress distribution around the body induces this yaw motion.

8:39AM A15.00004 Employing an Internal Wavemaker to Simulate Sensory Cues in the Plankton , A.C. TRUE, D.R. WEBSTER, M.J. WEISSBURG, J. YEN, Georgia Tech — Internal waves are a ubiquitous feature in many coastal marine ecosystems and as such are important features to consider in the spatiotemporal dynamics of thin planktonic layers. Oscillations of the pycnocline in stratified waters due to internal wave propagation generate fluxes of quantities, such as fluid momentum, thermal energy, and chemical concentration. These fields compose a set of hydrodynamic and thermochemical sensory cues that are fundamental to many planktonic life processes, including prey and predator detection, mate-tracking, habitat partitioning, nutrient and waste transport processes, and chemical communications. Thus, we expect that internal waves generate sensory cues in the water column, influence many fundamental biological processes, and broadly affect spatiotemporal productivity dynamics through unique biophysical coupling over a wide range of relevant scales. We constructed an internal wave generator facility to mimic characteristics that plankton observe *in situ*. Simultaneous particle image velocimetry (PIV) and laser-induced fluorescence (LIF) are employed on internal waves generated in a two-layer stratification to quantify wave-induced scalar fluxes. Difficulties inherent in scaling down in situ conditions for laboratory-scale behavioral assays are discussed in the context of accurately matching spatiotemporal scales from a planktonic point of view. Finally, the results are interpreted in the context of zooplankton sensory ecology.

8:52AM A15.00005 High Speed Tomographic PIV Measurements of Copepod Escape Jumps , D.R. WEBSTER, D.W. MURPHY, J. YEN, Georgia Tech — Copepods flee from predators via high-acceleration escape jumps that may reach speeds of up to 500 body lengths per second, i.e., relative speeds that are not reached by any other organism. We present time-resolved tomographic PIV measurements of the flow around an escaping calanoid copepod (*Calanus finmarchicus*). Persistent body and wake vortices are created by the impulsive momentum transfer to the fluid surrounding the animal. It is shown that an impulsive stresslet model better describes the flow than an impulsive Stokeslet. Azimuthal asymmetry of the strength and position of the wake vortex is analyzed and attributed to the strong ventral flows created by the metachronally beating swimming legs and to yawing of the body. In addition, the energy required by a copepod escape jump is estimated by calculating the viscous energy dissipation rate using the spatial gradients of the measured three-dimensional velocity field. Finally, the three-dimensional flow measurements are compared to previous axisymmetric CFD simulations.

9:05AM A15.00006 High-Speed Hopping: Time-Resolved Tomographic PIV Measurements of Water Flea Swimming , D.W. MURPHY, D.R. WEBSTER, J. YEN, Georgia Tech — Daphniids, also known as water fleas, are small, freshwater crustaceans that live in a low-to-intermediate Reynolds number regime. These plankters are equipped with a pair of branched, setae-bearing antennae that they beat to impulsively propel themselves, or "hop," through the water. A typical hop carries the daphniid one body length forward and is followed by a period of sinking. We present time-resolved tomographic PIV measurements of swimming by *Daphnia magna*. The body kinematics and flow physics of the daphniid hop are quantified. It is shown that the flow generated by each stroking antenna resembles an asymmetric viscous vortex ring. It is proposed that the flow produced by the daphniid hop can be modeled as a double Stokeslet consisting of two impulsively applied point forces separated by the animal width. The flow physics are discussed in the context of other species operating in the same Reynolds number range of 10 to 100: sea butterfly swimming and flight by the smallest flying insects.

9:18AM A15.00007 Ontogenetic propulsive transitions from viscous to inertial flow regimes in the medusae *Sarsia tubulosa* , KAKANI KATIJA, HOUSHUO JIANG, Applied Ocean Physics and Engineering, Woods Hole Oceanographic Institution, SEAN COLIN, Environmental Science, Roger Williams University, JOHN COSTELLO, Biology, Providence College — Among marine organisms, the influences of flow regimes on swimming strategies are largely unknown. As an approach to examine this issue, we quantified how transitions from viscous to inertially dominated flow regimes, which commonly occur during the development of marine animals, relate to changes in swimming strategies. We used the hydromedusae *Sarsia tubulosa* as a model organism for this investigation because its morphology and propulsive actuation mechanism are radially symmetric. This feature allows for determination of three-dimensional fluid quantities from two-dimensional flow measurement techniques. Digital particle image velocimetry was used to quantify the flow fields created by free-swimming hydromedusae and calculate the impulse generated by their swimming pulses at different life stages. Swimming strategies were evaluated by quantifying the relationship between impulse production and hydrodynamic swimming efficiency. Utilizing these metrics enable us to generalize our findings to the swimming strategies of other aquatic animals that swim in similar fluid regimes.

9:31AM A15.00008 3D-PTV measurement of the phototactic movement of algae in shear flow , TATSUYUKI MAEDA, TAKUJI ISHIKAWA, University of Tohoku, Department of Bioengineering and Robotics, HIRONORI UENO, KEIKO NUMAYAMA-TSURUTA, University of Tohoku, Department of Biomedical engineering, YOSUKE IMAI, University of Tohoku, Department of Bioengineering and Robotics, TAKAMI YAMAGUCHI, University of Tohoku, Department of Biomedical engineering — Recently, swimming motion of algae cells is researched actively, because algae fuel is one of the hottest topic in engineering. It is known that algae swim toward the light for photosynthesis however, the effect of a background flow on the unidirectional swimming is unclear. In this study, we used *Volvox* as a model alga and placed them in a simple shear flow with or without light stimulus. The shear flow was generated by moving two flat sheets in the opposite direction tangentially. A red LED light (wave length 660 nm) was used as an observation light source, and a white LED light was used to stimulate cells for the phototaxis. The trajectories of individual cells were measured by a 3D-PTV system, consists of a pair of high-speed camera with macro lenses. The results were analyzed to understand the effect of the background shear flow on the phototaxis of cells.

Sunday, November 18, 2012 8:00AM - 9:31AM –

Session A16 Biofluids: Valves, Stents and Devices 28B - Chair: Marija Vukicevic, Clemson University

8:00AM A16.00001 Transcatheter valve implantation can alter fluid flow fields in aortic sinuses and ascending aorta , NEELAKANTAN SAIKRISHNAN, AJIT YOGANATHAN, Georgia Institute of Technology — Transcatheter aortic valves (TAVs) are valve replacements used to treat aortic stenosis. Currently, these have been used in elderly patients at high-risk for open-heart procedures. Since these devices are implanted under fluoroscopic guidance, the implantation position of the valve can vary with respect to the native aortic valve annulus. The current study characterizes the altered hemodynamics in the aortic sinus and ascending aorta under different implantation (high and low) and cardiac output (2.5 and 5.0 L/min) conditions. Two commonly used TAV designs are studied using 2-D Particle Image Velocimetry (PIV). 200 phase locked images are obtained at every 25ms in the cardiac cycle, and the resulting vector fields are ensemble averaged. High implantation of the TAV with respect to the annulus causes weaker sinus washout and weaker sinus vortex formation. Additionally, the longer TAV leaflets can also result in a weaker sinus vortex. The level of turbulent fluctuations in the ascending aorta did not appear to be affected by axial positioning of the valve, but varied with cardiac output. The results of this study indicates that TAV positioning is important to be considered clinically, since this can affect coronary perfusion and potential flow stagnation near the valve.

8:13AM A16.00002 Integrating bio-prosthetic valves in the Fontan operation - Novel treatment to control retrograde flow in caval veins[1] , MARIJA VUKICEVIC, TIMOTHY CONOVER, JIAN ZHOU, Clemson University, TAIN-YEN HSIA, Great Ormond Street Hospital, London, UK, RICHARD FIGLIOLA, Clemson University — For a child born with only one functional heart ventricle, the sequence of palliative surgeries typically culminates in the Fontan operation. This procedure is usually successful initially, but leads to later complications, for reasons not fully understood. Examples are respiratory-dependent retrograde flows in the caval and hepatic veins, and increased pulmonary vascular resistance (PVR), hypothesized to be responsible for elevated pressure in the liver and disease of the liver and intestines. Here we study the parameters responsible for retrograde flows in the inferior vena cava (IVC) and hepatic vein (HV), and investigate two novel interventions to control retrograde flow: implanting either a Medtronic Contegra valved conduit or an Edwards lifescience pericardial aortic valve in the IVC or HV. We performed the experiments in a multi-scale, patient specific mock circuit, with normal and elevated PVR, towards the optimization of the Fontan circulation. The results show that both valves can significantly reduce retrograde flows in the veins, suggesting potential advantages in the treatment of the patients with congenital heart diseases.

[1]Fondation Leducq

8:26AM A16.00003 Finite Element CURVIB method for fluid-structure interaction simulations of tissue heart valves , ANVAR GILMANOV, HENRYK STOLARSKI, TRUNG LE, FOTIS SOTIROPOULOS, University of Minnesota — A new fluid-structure interaction (FSI) model is developed for solving the three-dimensional unsteady Navier-Stokes equations in domains with arbitrarily complex tissues undergoing large deformation. The method employs the sharp-interface CURVIB method [L. Ge, F. Sotiropoulos, JCP, 225 (2007) 1782-1809] for handling complex moving boundaries, with a finite element (FE) model, which can handle large structural/tissue deformations using the nonlinear Kirckhhoff thin shells theory. A new treatment of the flexible immersed body is introduced to handle the thin body surfaces. A version of Aitken's acceleration for strong fluid-structure coupling is used to calculate the responses of the flexible bodies to the applied fluid load. The new, rotation-free finite element formulation for large deformations of thin shells has been extensively tested and validated for a range of relevant problems. The coupled FE-CURVIB FSI model is validated for vortex induced oscillations of a flexible cantilever and applied to simulate physiologic pulsatile flows through a tissue aortic heart valve in an anatomic artery geometry. The results show the good convergence property of the new FSI algorithm and demonstrate its promise for a broad range of biological applications.

8:39AM A16.00004 Experimental investigation of the effects of inserting a bovine venous valve in the inferior vena cava of Fontan circulation , ARVIND SANTHANAKRISHNAN, JACOB JOHNSON, MONICA KOTZ, ELAINE TANG, REZA KHIABANI, AJIT YOGANATHAN, Wallace H. Coulter Department of Biomedical Engineering, Georgia Institute of Technology & Emory University, KEVIN MAHER, Sibley Heart Center Cardiology, Children's Healthcare of Atlanta at Egleston — The Fontan procedure is a palliative surgery performed on patients with single ventricle (SV) congenital heart defects. The SV is used for systemic circulation and the venous return from the inferior vena cava (IVC) and superior vena cava (SVC) is routed to the pulmonary arteries (PA), resulting in a total cavopulmonary connection (TCPC). Hepatic venous hypertension is commonly manifested in the Fontan circulation, leading to long-term complications including liver congestion and cirrhosis. Respiratory intrathoracic pressure changes affect the venous return from the IVC to the PA. Using a physical model of an idealized TCPC, we examine placement of a unidirectional bovine venous valve within the IVC as a method of alleviating hepatic venous hypertension. A piston pump is used to provide pulsatility in the internal flow through the TCPC, while intrathoracic pressure fluctuations are imposed on the external walls of the model using a pair of linear actuators. When implanted in the extrathoracic position, the hepatic venous pressure is lowered from baseline condition. The effects of changing caval flow distribution and intrathoracic pressure on TCPC hemodynamics will be examined.

8:52AM A16.00005 Mitigation of Shear-Induced Blood Damage of Mechanical Bileaflet Heart Valves using Embedded Vortex Generators[1] , PABLO HIDALGO, SIVAKKUMAR ARJUNON, NEELAKANTAN SAIKRISHNAN, AJIT YOGANATHAN, ARI GLEZER, Georgia Institute of Technology — The strong transitory shear stress generated during the time-periodic closing of the mechanical prosthetic bileaflet aortic heart valve, is considered to be one of the main factors responsible for complications, associated with thrombosis and thromboembolism. These flow transients are investigated using phase and time-averaged PIV in a low-volume (about 150 ml) test setup that simulates the pulsatile physiological conditions associated with a 23 mm St. Jude Medical valve. The PIV measurements are accompanied by continuous monitoring of the ventricular and aortic pressures and valve flow rate. Following the valve closure, the leakage flow between the valve leaflets is caused by the pressure buildup across the leaflets, leading to the formation of a regurgitation jet starting from the BMHV B-datum line. As in a typical starting jet, a counter-rotating vortex pair is formed along each leaflet edge and the vorticity sheet is associated with high shear stress that may be result in blood platelet activation. The present investigation demonstrates that the placement of arrays of mm-scale vortex generators near the edges of the leaflets diffuses the vortex sheet and suppresses the formation of these vortices, weakening the local velocity gradients and small-scale vortical structures.

[1]Supported by NIH and NSF.

9:05AM A16.00006 Simulations of pulsatile suspension flow through bileaflet mechanical heart valves to quantify platelet damage , BRIAN YUN, CYRUS AIDUN, AJIT YOGANATHAN, Georgia Institute of Technology — Studies have shown that high shear stress and long exposure times on platelets have a strong impact on thromboembolic complications in bileaflet mechanical heart valves (BMHVs). This numerical study quantifies the platelet damage incurred in pulsatile flow through various BMHV designs. The lattice-Boltzmann method with external boundary force (LBM-EBF) was implemented to simulate pulsatile flow and capture the dynamics and surface shear stresses of modeled platelets with realistic geometry. The platelets are released in key regions of interest in the geometry as well as at various times of the cardiac cycle. The platelet damage is quantified using a linear shear stress-exposure time blood damage index (BDI) model. The multiscale computational method used to quantitatively measure the BDI during the pulsatile flow has been validated as being able to accurately capture bulk BMHV fluid flow and for accurately quantifying platelet damage in BMHV flows. These simulations will further knowledge of the geometric features and cardiac cycle times that most affect platelet damage. This study will ultimately lead to optimization of BMHV design in order to minimize thromboembolic complications.

9:18AM A16.00007 Bio-inspired, low-cost, self-regulating valves for drip irrigation in developing countries , PAWEL ZIMOCH, ELIOTT TIXIER, ANETTE HOSOI, AMOS WINTER, Massachusetts Institute of Technology — We use nonlinear behavior of thin-walled structures - an approach inspired by biological systems (the human airway, for example) - to address one of the most important problems facing subsistence farmers in developing countries : lack of access to inexpensive, water-efficient irrigation systems. An effective way of delivering water to crops is through a network of drippers, with up to 85% of the water delivered being absorbed by plants. However, of the 140m hectares of cropped land in India, only 61m are irrigated and just 5m through drip irrigation. This is, in part, due to the relatively high cost of drip irrigation. The main cost comes from the requirement to pump the water at relatively high pressure (>1bar), to minimize the effect of uneven terrain and viscous losses in the network, and ensure that each plant receives the same amount of water. We demonstrate that the pressure required to drive the system can be reduced significantly by using thin-walled structures to design drippers with completely passive self-regulation that activates at approximately 0.1bar. We also report on our work towards reducing the overall price of drip irrigation systems by as much as 90%, and making them more affordable for 800 million subsistence farmers worldwide.

Sunday, November 18, 2012 8:00AM - 9:44AM –

Session A17 Biofluids: Flapping and Flying I 28C - Chair: Haecheon Choi, Seoul National University

8:00AM A17.00001 Flying in Two Dimensions , MANU PRAKASH, THIBAUT BARDON[1], Stanford University — It has long been proposed that insect flight might have evolved on a fluid interface. Surface of a pond provides an ecological niche which is exploited by a large number of species capable of locomotion on a fluid interface. Here we describe the discovery of constrained flight in two dimensions as a novel mode of locomotion used by water lily beetles (genus Galerucella). Because water lily beetles are also capable of three-dimensional free flight, this novel two-dimensional locomotion provides us with a unique model system to explore both the transition between two and three dimensional flight and the associated energetics. Here we present a comparative analysis of this transition in terms of wing stroke angles associated with two and three dimensional flight, as well as modeling surface tension forces on both the horizontal and vertical axes. Special attention is paid to the dynamics and energetics of flight in two-dimensions, focusing on the interaction of the wing strokes with the fluid interface and the capillary-gravity wave drag associated with two-dimensional propulsion.

[1]Current Address: Ecole Polytechnique, France

8:13AM A17.00002 Axial flow effects on robustness of vortical structures about actively deflected wings in flapping flight[1] , ALBERT MEDINA, University of California, Los Angeles, JIHOON KWEON, HAECHEON CHOI, Seoul National University, JEFF D. ELDREDGE, University of California, Los Angeles — Flapping wing flight has garnered much attention in the past decade driven by our desire to understand capabilities observed in nature and to develop agile small-scale aerial vehicles. Nature has demonstrated the breadth of maneuverability achievable by flapping wing flight. However, despite recent advances the role of wing flexibility remains poorly understood. In an effort to develop a deeper understanding of wing deflection effects and to explore novel approaches to increasing leading-edge vortex robustness, this three-dimensional computational study explores the aerodynamics of low aspect ratio plates, in hovering kinematics, with isolated flexion lines undergoing prescribed deflection. Major flexion lines, recognized as the primary avenue for deflection in biological fliers, are isolated here in two distinct configurations, resulting in deflection about the wing root and the wing tip, respectively. Of interest is the interaction between axial flow along the span and the vortical structures about the wing. It is proposed that the modes of deflection explored may provide a means of axial flow control for favorably promoting LEV robustness over a broad range of flapping conditions, and provide insight into the nature of flexibility in flapping wing flight.

[1]National Science Foundation, National Research Foundation of Korea

8:26AM A17.00003 Quasi-Steady Limit of Flow Structure on Flapping Wing in Mean Flow , MATTHEW BROSS, CEM OZEN, DONALD ROCKWELL, Lehigh University — A limiting case of flapping motion of a wing (low aspect ratio plate) in presence of incident steady flow is compared to a rotating wing in quiescent fluid, in order to clarify the effect of advance ratio J (ratio of free-stream velocity to tangential velocity of wing) on the structure of the leading-edge vortex. Stereoscopic particle image velocimetry leads to patterns of vorticity, velocity contours, and streamlines. For each value of J, the effective angle of attack is held constant at 45°, while the wing rotates from rest through 270°. While at rest, the wing at high angle of attack in the presence of a steady free-stream gives rise to fully stalled flow. After the onset of rotation, the fully stalled region very quickly transforms to a stable leading edge vortex. Despite the change in advance ratio, the development of the flow structure around the wing throughout the rotation maneuver is similar, especially in the leading edge vortex region, as evidenced by patterns of streamline topology. To further demonstrate the effect of J, three-dimensional representations of of spanwise-oriented vorticity, spanwise velocity, and Q were constructed for hovering flight and forward flight.

8:39AM A17.00004 Inline Motion in Flapping Foils for Improved Force Vectoring Performance , JACOB IZRAELEVITZ, Massachusetts Institute of Technology, GABRIEL WEYMOUTH, Singapore-MIT Alliance for Research and Technology, MICHAEL TRIANTAFYLLOU, Massachusetts Institute of Technology — Flapping foils are a promising alternative actuation technique for aerial and underwater vehicles because they can drastically improve maneuverability by vectoring the actuator force. However, the standard implementation of a flapping foil motion, where the foil is oscillated exactly perpendicular to the free stream flow, does not fully develop this force vectoring capability. Many biological examples of flapping foil actuators include an additional degree of freedom, where the foil is allowed to translate parallel to the flow. This degree of freedom can either powerfully augment the mean lift, or mitigate oscillating lift forces for improved thrust efficiency. We develop a parameterization of this inline motion and outline various motion schemes to improve the force vectoring performance of a flapping foil actuator. We then investigate these motion schemes with both CFD solutions and towing tank experiments, thereby expanding the force vectoring options available for the flapping foil actuator.

8:52AM A17.00005 Three-dimensional flow measurements of a differentially driven flapping wing mechanism[1] , ERIC HARDESTER, SCOTT THOMSON, TADD TRUSCOTT, Brigham Young University — We present the results of a 3D visualization of the flow field around a differentially driven model of a ladybug wing using Synthetic Aperture PIV (SAPIV) at positions above, below, and 1 chord length behind the wing. Developments in the micro air vehicle field (chord length $<$ 15 cm) have shown advantages in stability and lift generation for flapping wings over fixed wings. These advantages are believed to come from the increased lift caused by various flow structures such as Leading Edge Vortices (LEVs) created by flapping wings, and a "draining" process through the core. Visualizations and analysis of the wake structures of flapping wings have been made using Particle Image Velocimetry (PIV) techniques, allowing 2-dimensional slices of the flow to be analyzed. However, the wake structures of flapping wings are 3-dimensional, making traditional PIV techniques inadequate for full visualization of the wake structure. SAPIV provides a method for gathering 3-dimensional flow measurements of the flow around the model of the ladybug wing. Focus is placed on the development of the LEV and the draining process through the core.

[1]AFOSR Grant FA9550-10-0334

9:05AM A17.00006 Flight simulations of a two-dimensional flapping wing by the IB-LBM , TAKAJI INAMURO, YUSUKE KIMURA, KOSUKE SUZUKI, Department of Aeronautics and Astronautics, Kyoto University — Two-dimensional symmetric flapping flight is investigated by the immersed boundary-lattice Boltzmann method. First, we investigate the effect of the Reynolds number on flows around symmetric flapping wings under no-gravity field and find that for high Reynolds numbers ($Re \geq 55$) asymmetric vortices with respect to the horizontal line appear and the time-averaged lift force is induced on the wings. Secondly, we study the motion of the model with an initial rotational disturbance and find that the motion is rotationally unstable. That is, once the model starts rotating, the rotational motion rapidly increases due to a complicated vorticity field around the wings. Finally, we propose a simple way to control the rotational and horizontal motion by bending and flapping the tip of the wing. With the control we can achieve an upward stable motion in spite of the complicated vorticity field around the wings.

9:18AM A17.00007 Numerical modeling of flexible insect wings using volume penalization , THOMAS ENGELS, M2P2-CNRS and CMI, Aix-Marseille University, Marseille, France & Institut fuer Stroemungmechanik und Technische Akustik (ISTA), TU Berlin, Germany, DMITRY KOLOMENSKIY, Centre Europeen de Recherche et de Formation Avancee en Calcul Scientifique (CERFACS), Toulouse, France, KAI SCHNEIDER, M2P2-CNRS and CMI, Aix-Marseille University, Marseille, France, JOERN SESTERHENN, Institut fuer Stroemungmechanik und Technische Akustik (ISTA), TU Berlin, Germany — We consider the effects of chordwise flexibility on the aerodynamic performance of insect flapping wings. We developed a numerical method for modeling viscous fluid flows past moving deformable foils. It extends on the previously reported model for flows past moving rigid wings (J Comput Phys 228, 2009). The two-dimensional Navier-Stokes equations are solved using a Fourier pseudo-spectral method with the no-slip boundary conditions imposed by the volume penalization method. The deformable wing section is modeled using a non-linear beam equation. We performed numerical simulations of heaving flexible plates. The results showed that the optimal stroke frequency, which maximizes the mean thrust, is lower than the resonant frequency, in agreement with the experiments by Ramananarivo et el. (PNAS 108(15), 2011). The oscillatory part of the force only increases in amplitude when the frequency increases, and at the optimal frequency it is about 3 times larger than the mean force. We also study aerodynamic interactions between two heaving flexible foils. This flow configuration corresponds to the wings of dragonflies. We explore the effects of the phase difference and spacing between the fore- and hind-wing.

9:31AM A17.00008 A flight control through unstable flapping flight[1] , MAKOTO IIMA, Hiroshima University, NAOTO YOKOYAMA, Kyoto University, NORIO HIRAI, Osaka Prefecture University, KEI SENDA, Kyoto University — We have studied a flight control in a two-dimensional flapping flight model for insects. In this model, the model of center-of-mass can move in both horizontal and vertical directions according to the hydrodynamic force generated by flapping. Under steady flapping, the model converges to steady flight states depending on initial conditions. We demonstrate that simple changes in flapping motion, a finite-time stop of flapping, results in changes in the vortex structures, and the separation of two steady flight state by a quasi-steady flight. The model's flight finally converges to one of the final states by way of the quasi-steady state, which is not observed as a (stable) steady flight. The flight dynamic has been also analyzed.

[1]KAKENHI (23540433, 22360105, 21340019) and CREST No. PJ74100011

Sunday, November 18, 2012 8:00AM - 9:44AM –

Session A18 Biofluids: General 28D - Chair: Luca Brandt, KTH, Stockholm

8:00AM A18.00001 Fluid fragmentation and disease transmission , LYDIA BOUROUIBA, JOHN W.M. BUSH, Massachusetts Institute of Technology — The transfer of pathogens from infected to non-infected members of a population is critical in determining the outcome of an epidemic. However, fundamental mechanisms of pathogen spreading remain poorly understood. We here present the results of combined experimental and theoretical studies of the role of fluid fragmentation in the transmission of a number of common pathogens, with a particular focus on those causing respiratory infections.

8:13AM A18.00002 The fluid dynamics of human birth , ANDREA LEHN, MEGAN C. LEFTWICH, The George Washington University — This study investigates the fluid dynamics associated with the human birth process. Specifically, we investigate the role of the viscosity of the amniotic fluid in transferring force from the contracting uterus to the fetus during delivery. This experimental work uses an approximate uterus and dilated cervix–fabricated with liquid latex–filled with a fluid of known viscosity and an oblong solid fetus. The force required to extract the fetus is recorded for several values of amniotic viscosity. The study looks at both pull-out force values (where the fetus is pulled from outside the uterus) and push-out force values (where pressure in the experimental uterus is used to remove the fetus). In addition to the viscosity study, we also investigate the increased force required to deliver an offset fetus by tilting the major axis of the oblong fetus and repeating the pull-and push-out experiments. This study will provide knowledge about the fundamental fluid dynamic processes involved in human birth.

8:26AM A18.00003 Evolution of a pre and post lens tear film with a contact lens , MATTHEW GERHART, DANIEL ANDERSON, George Mason University — The work is the development, implementation, and analysis of a two-dimensional tear film model including a porous contact lens. The geometry of the problem is: a pre-lens layer that is a thin tear film between the outside air and contact lens, a contact lens that is a rigid but movable porous substrate, and a post-lens layer that is a thin film layer between the contact lens and the cornea. We are looking at short and long term behavior of the evolution of the thin film in the pre-lens layer coupled with the porous layer and the thin squeeze film in the post-lens layer. We model the different behaviors that arise as the Darcy number, evaporation effects, and boundary flux conditions change.

8:39AM A18.00004 A Model Problem for Tear Film Distribution on a Moving Rectangular Domain , QUAN DENG, TOBIN DRISCOLL, RICHARD BRAUN, University of Delaware — We consider a model problem for the pre-corneal tear film on a moving 2D rectangular domain. The problem considers a thin Newtonian layer covered by an insoluble surfactant representing the effect of polar lipids. A non-linear PDE for film thickness from the lubrication approximation, together with a nonlinear PDE for the surfactant concentration is solved using the method of lines with spectral methods in space. The Marangoni effect couples the variables together. Numerical experiments using different end motions (realistic or sinusoidal) and perturbations to the surfactant distributions (to imitate observe lipid distributions *in vivo*) were performed. The results indicate that some *in vivo* elements of the tear film distribution with relatively long length scales are captured by the model, but some fine-scale phenomena are not captured. If time permits, results from a two layer model will presented.

8:52AM A18.00005 Modeling Tear Film Dynamics on a 2-D Eye-shaped Domain , LONGFEI LI, RICHARD BRAUN, University of Delaware, KARA MAKI, Rochester Institute of Technology, WILLIAM HENSHAW, Lawrence Livermore National Laboratory — We study tear film dynamics on a 2-D eye-shaped domain using a lubrication model. Time dependent flux boundary conditions that model the lacrimal gland tear supply and punctal drainage are imposed. We solved the model equations with Overture computational framework. Results reveals our model captures the hydraulic connectivity and other key physics of human tear film observed *in vivo*. Comparisons are made with existing models and experiments. Should time permit, osmolarity dynamics (salt ion concentration) will be included.

9:05AM A18.00006 Two Layer Model for Local Tear Film Dynamics , NICHOLAS GEWECKE, RICH BRAUN, University of Delaware, CHRIS BREWARD, University of Oxford, EWEN KING-SMITH, The Ohio State University — Many tear film models utilize a single-layer approach that represents only the aqueous layer, which constitutes the majority of the tear film. In such models, the layer is dominated by shear stresses. Some recent models have incorporated surfactant effects at the liquid-air interface to model the effects of polar lipids there. Clinical observations of the lipid layer indicate more complicated dynamics of the lipid layer than demonstrated by these previous models. The model presented in this talk includes a thin lipid layer between the aqueous layer and the air, which is treated as an extensional flow. Our results demonstrate formation of lipid drops, with the number of drops dependent upon the parameters of the system, especially the thickness ratio between the lipid and aqueous layers.

9:18AM A18.00007 On conjoining pressures in the tear film , JAVED SIDDIQUE, Penn State York, NICHOLAS GEWECKE, RICH BRAUN, University of Delaware — We study the local tear film dynamics in a two-layer model with a Newtonian extensional layer over a Newtonian shear layer with a surfactant between. The upper layer represents the lipid layer and the underlying layer the aqueous layer in the tear film. We study the effect of the ions on the conjoining pressure in the aqueous layer using a Debye-Huckel approximation. If time permits, we will treat the evaporation of the water from the underlying aqueous layer and the effect of increasing osmolarity of the aqueous and the interaction with the conjoining pressure. More complicated conjoining pressure contributions are added as needed.

9:31AM A18.00008 Convective transport resistance in the vitreous humor , ANITA PENKOVA, SATWINDAR SADHAL, KOMSAN RATANAKIJSUNTORN, University of Southern California, REX MOATS, YANG TANG, Children's Hospital Los Angeles, PATRICK HUGHES, MICHAEL ROBINSON, SUSAN LEE, Allergan, Inc — It has been established by MRI visualization experiments that the convection of nanoparticles and large molecules with high rate of water flow in the vitreous humor will experience resistance, depending on the respective permeabilities of the injected solute. A set of experiments conducted with Gd-DTPA (Magnevist, Bayer AG, Leverkusen, Germany) and 30 nm gadolinium-based particles (Gado CELLTrack^{TM}, Biopal, Worcester, MA) as MRI contrast agents showed that the degree of convective transport in this Darcy-type porous medium varies between the two solutes. These experiments consisted of injecting a mixture of the two (a 30 μl solution of 2% Magnevist and 1% nanoparticles) at the middle of the vitreous of an ex vivo whole bovine eye and subjecting the vitreous to water flow rate of 100 μl/min. The water (0.9% saline solution) was injected at the top of the eye, and was allowed to drain through small slits cut at the bottom of the eyeball. After 50 minutes of pumping, MRI images showed that the water flow carried the Gd-DTPA farther than the nanoparticles, even though the two solutes, being mixed, were subjected to the same convective flow conditions. We find that the convected solute lags the water flow, depending on the solute permeability. The usual convection term needs to be adjusted to allow for the filtration effect on the larger particles in the form (1-σ)$\mathbf{u}\cdot\nabla c$ with important implications for the modeling of such systems.

Sunday, November 18, 2012 8:00AM - 9:44AM –

Session A19 Mini-Symposium: High-Speed, High-Energy Multimaterial Flows 28E - Chair: Gustaaf Jacobs, San Diego State University

8:00AM A19.00001 Development of a multiscale Eulerian-Lagrangian method for high-speed multi-material flows , UDAY KUMAR, University of Iowa — Explosions and combustion processes generate environments where fluid turbulence and shocks have an intimate and mutual interaction with solid materials. While there has been considerable effort to determine suitable models for low speed (i.e. incompressible and low subsonic) flows and in solid mechanics, there is sparse work of this nature in the context of compressible (particularly shocked) multi-material flows. We seek to automate the development of particle transport models and closure by employing a multi-scale approach. In order to develop multi-scale modeling capabilities for high-speed particle-laden flows, we will discuss a joint effort that brings together a macro-scale, hig–order resolution Eulerian-Lagrangian method, and a micro-scale, full-resolution, high-fidelity first principle model for direct numerical simulations of shocked flows through heterogeneous media and micro-macro coupling. The multiscale approach is constructed in a hierarchical framework where high-fidelity DNS of particle interactions with shocked flows are employed at the micro-scale with full description of flow around resolved particles. At the macroscale the particles are modeled using Lagrangian point cloud methods. The linkage between scales is established through artificial neural networks (ANNs) trained to assimilate micro-scale physics and serve as closure models for the macro-scale simulations.

8:26AM A19.00002 Toward Rigorous Modeling of Extreme Compressible Multiphase Flows[1] , Y. LING, M. PARMAR, S. ANNAMALAI, S. BALACHANDAR, University of Florida, D.L. FROST, Mcgill University — Modeling is an important approach to investigate extreme compressible multiphase flows, such as explosive dispersal of particles/droplets and volcanic eruptions. Since the scale of practical interest is much larger than the particle size, point-particle models are usually employed in macro-scale simulations. We have developed a physics-based point-particle model, which divides the overall particle force and heat transfer into physically meaningful contributions. The effects of finite Reynolds and Mach numbers and finite particle volume fraction on each force and heat transfer contribution are incorporated. The model is used to simulate the problems of a planar shock wave interacting with a dense particle curtain and the rapid radial spreading of an annular bed of particles emplaced around a cylindrical explosive. In both cases the numerical results agree well with experimental data. Shock refractions that occur at the particle fronts generate reflected and transmitted shocks. These shocks introduce strong velocity and pressure gradients across the curtain, which causes the curtain to expand. The presentation will also discuss the modeling and simulation challenges of detonation-particle interaction and strong interparticle interaction arising from large volume fraction.

[1]Supported by AFOSR (FA 9550-10- 1-0309).

8:52AM A19.00003 Closure Models for Turbulent Particle-laden Flows from Particle-resolved Direct Numerical Simulation[1] , SHANKAR SUBRAMANIAM, SUDHEER TENNETI, MOHAMMAD MEHRABADI, Iowa State University, RAHUL GARG, National Energy Technology Laboratory — Gas-phase velocity fluctuations in fixed particle beds and freely evolving suspensions are quantified using a particle-resolved direct numerical simulation (PR-DNS). The flow regime corresponds to gas-solid systems typically encountered in fluidized bed risers, with high solid to gas density ratio and particle diameter being greater than the dissipative length scales. The kinetic energy associated with gas-phase velocity fluctuations in homogeneous monodisperse fixed beds is characterized as a function of solid volume fraction ϕ and the Reynolds number based on the mean slip velocity Re. A simple scaling analysis is used to explain the dependence of k on ϵ and Re. The steady value of k results from the balance between the source of k due to interphase transfer of kinetic energy, and the dissipation rate (ϵ) of k in the gas-phase. It is found that the dissipation rate of k in gas-solid flows can be modeled using a length scale that is analogous to the Taylor microscale used in single-phase turbulence. Using the PR-DNS data for k and ϵ we also infer an eddy viscosity for gas-solid flow. For the parameter values considered here, the level of gas-phase velocity fluctuations in freely evolving suspensions differs by only 10% from the value for the corresponding fixed beds.

[1]Funded in part by the US Department of Energy's National Energy Technology Laboratory Grant DE-FC26-07NT43098 (Advanced Research) and the National Science Foundation's grant CBET 1134500.

9:18AM A19.00004 Modeling Detonation of Heterogeneous Explosives with Embedded Inert Particles Using Detonation Shock Dynamics: Normal and Divergent Propagation in Regular and Simplified Microstructure , SCOTT STEWART, University of Illinois at Urbana-Champaign — We use a detonation shock propagation model, Detonation Shock Dynamics(DSD) to compute the interaction of a detonation shock wave that passes over a series of inert spherical particles embedded in a high explosive material. DSD provides an efficient means to study the dynamics of lead shock waves without the necessity of simulating the entire multi-material, reactive flow field. We derive partial differential equations for the motion of a detonation shock that obeys a linear shock normal velocity-curvature relation in a cylindrical coordinate system and in a moving, shock-attached coordinate system. The shock dynamics equations are solved numerically, in a unit-cell configuration. We describe the short-term and long-term behavior of the shock wave as it passes over the particles. We describe both the averages and character of the stochastic behavior that affects long-term average properties for microstructure in which the inert particles are periodically and randomly spaced.

Sunday, November 18, 2012 8:00AM - 9:44AM –

Session A20 Turbulent Boundary Layers I: Large Structures 30A - Chair: Nicholas Hutchins, University of Melbourne

8:00AM A20.00001 The large-scale wall-to-wall interaction in fully developed turbulent channel flow , YONG SEOK KWON, KAPIL CHAUHAN, CHARITHA DE SILVA, JASON MONTY, NICHOLAS HUTCHINS, The University of Melbourne — The geometry and boundary conditions in a turbulent channel flow ensure symmetry or anti-symmetry of the time-averaged turbulence statistics across the channel half-height ($y = h$). These statistics are conventionally studied over one half of the channel. However instantaneous fluctuating velocity fields often show coherent large-scale anti-symmetric features on either side of the centreline. This talk will present the interaction of such large-scale events with the turbulence originating on the opposite wall. Particle Image Velocimetry (PIV) is utilized to obtain two-component flow fields across the full channel height for $Re_\tau = hu_\tau/\nu \simeq 1000 - 4000$. Ensemble averages of fluctuating velocity fields conditioned on clockwise and counter-clockwise swirl events at reference wall-normal positions ranging from $y = 0.1h$ to $y = h$ reveal the presence of the wall-to-wall interaction of large-scale motions. Further results from conditional averaging such as fluctuating velocity contour plots and vector fields will be presented to show this wall-to-wall interaction in channel flow. A decomposition of flow fields into symmetric and anti-symmetric components shows that the anti-symmetry predominantly exists for very large-scale motions (VLSMs) of the order $10h$.

8:13AM A20.00002 Experimental investigation of coherent structures in turbulent pipe flow using a large-scale pipe flow facility , DAVID DENNIS, University of Liverpool — In recent years it has been shown by various researchers, using either experimental techniques or direct numerical simulations, that coherent structures (i.e. features of the flow that persist in space and time) such as hairpin vortices, vortex packets, and very large scale motions (or superstructures) play an important role in wall-bounded turbulent flows (boundary layers, pipes and channel flows). A large-scale recirculating pipe flow facility at the University of Liverpool has been developed to enable the investigation of large and very large scale coherent motions in turbulent pipe flow. The facility includes a 100mm-diameter working section, consisting of individual modules of precision-bore borosilicate glass tubes each 1.027m long, totalling 22 metres in length. Experimental measurements using high-speed stereoscopic particle image velocimetry at approximately 210 pipe diameters downstream of the inlet are made possible using a unique mechanical arrangement for performing the calibration. Reynolds numbers of up to $Re_D = 10^5$ can be reached when the working fluid is water.

8:26AM A20.00003 Study of very-large-scale motions in turbulent pipe flow[1] , JAE HWA LEE, HYUNG JIN SUNG, KAIST — Direct numerical simulation (DNS) of turbulent pipe flow was performed at Re_τ=544 to investigate the spatial organization of the very large-scale motions (VLSMs). The streamwise domain length employed here was $30R$, where R is the pipe radius. Inspection of the three-dimensional instantaneous fields showed that adjacent large-scale packet-like structures combine to form the VLSMs, and this formation process was attributed to continuous stretching of the hairpins coupled with lifting-up and backward curling of the vortices. To support our results found in the analysis of the instantaneous flow fields, we applied the spatial filter to decompose the signal into two length scales related to the VLSMs and smaller structures. The resulting streamwise length scale from the streamwise two-point correlations showed that the magnitude of the correlations for the VLSMs is larger than that from the large-scale motions (LSMs) through all directions. In addition, the mean inclination angle to the wall for the smaller scale structures was found to be larger than that of the VLSMs. These findings support the previous conjecture of Kim & Adrian (1999) that the coherent alignment of LSMs creates the VLSMs.

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

8:39AM A20.00004 Structural organization of large and very-large scales in turbulent pipe flow simulation[1] , JON BALTZER, RONALD ADRIAN, Arizona State University, XIAOHUA WU, Royal Military College of Canada — The physical structures of velocity are examined in a recent DNS of fully developed incompressible turbulent pipe flow at $Re_D = 24\,580$ ($R^+ = 684.8$) with a periodic domain length of 30 pipe radii R (Wu, Baltzer, & Adrian, J. Fluid Mech., 2012). In this simulation, the long motions of negative velocity fluctuation correspond to large fractions of energy present at very long streamwise wavelengths ($\geq 3R$). We study how long motions are composed of smaller motions. We characterize the spatial arrangements of very large scale motions (VLSMs) and find that they possess dominant helix angles (azimuthal inclinations relative to streamwise) that are revealed by 2D and 3D two-point spatial correlations of velocity. The correlations also reveal that the shorter, large scale motions (LSMs) that concatenate to comprise the VLSMs are themselves more streamwise aligned. We show that the largest VLSMs possess a form similar to roll cells and that they appear to play an important role in organizing the flow, while smaller scales of motion are necessary to create the strong streaks of velocity fluctuation that characterize the flow.

[1]Supported by NSF Award CBET-0933848.

8:52AM A20.00005 Amplitude modulation by a synthetic very large-scale motion (VLSM) in the turbulent boundary layer , I. JACOBI, B.J. MCKEON, Caltech — A flat-plate boundary layer is dynamically forced with a spatially-impulsive, two-dimensional roughness patch, introducing a periodic velocity disturbance. This synthetic very large-scale motion (VLSM) is isolated by a phase-locked decomposition of the velocity field, and its relationship with other scales in the flow is studied in the context of the apparent amplitude modulation of small-scale motions by large scales. The modulation effect is described as a phase-relationship between the different scales, where the phase difference is identified by cross-correlation. The extent to which the synthetic VLSM can be treated as a linear superposition on the base flow is investigated, and it is shown that the phase measurements can be used to describe the dominant large-scale motions in the unperturbed flow as well as the non-equilibrium distortion of the boundary layer by the roughness. Cospectral techniques are employed to identify the particular large-scale motions dominant in the amplitude modulation in both perturbed and unperturbed flows, and it is shown that VLSMs are significant to the modulation process, even in the unperturbed boundary layer. This study is supported by the Air Force Office of Scientific Research under grant #FA9550-09-1-0701(program manager John Schmisseur).

9:05AM A20.00006 Conditional analysis of the statistics near the turbulent/non-turbulent interface of turbulent boundary layers , HIROKI OGASAWARA, Nagoya University, TAKASHI ISHIHARA, Nagoya University, JST CREST — Direct numerical simulations of zero-pressure-gradient turbulent boundary layer (TBL) along a flat plate are used to investigate the properties of turbulent/non-turbulent interface of the TBL. The Reynolds numbers based on the momentum thickness are from 800 to 2200. The interface is defined as the set of the outermost points of the region with the absolute values of vorticity greater than $\alpha U_\infty/\delta$, where U_∞ is the free stream velocity, δ is the boundary layer thickness, and $\alpha = 0.3 - 0.7$ is used in our analysis. The analysis of conditional statistics of streamwise velocity u and spanwise vorticity ω_z near the interface shows that there is a shear (a sharp change of u) due to a sharp change of the distribution of vorticity ω_z near the interface. These results are consistent with the experiments by Semin et al. (2011). DNS data also show that the shear rate near the interface is larger when the height of the interface is smaller. Visualization of the time series of the interfaces suggests that the interfaces are moving downstream without remarkable changes in their shape during a time period of the order of δ/U_∞.

9:18AM A20.00007 The Turbulent/Non-turbulent Interface and Entrainment in a Boundary Layer , KAPIL CHAUHAN, JIMMY PHILIP, NICHOLAS HUTCHINS, CHARITHA DE SILVA, IVAN MARUSIC, The University of Melbourne — The turbulent/non-turbulent (T/NT) interface in a zero pressure gradient turbulent boundary layer ($Re_\tau \approx 8000$) is examined using particle image velocimetry. The interface is detected using local turbulent kinetic energy and proves to be an effective method for boundary layers. Statistically the interface exhibits a normal distribution characterizing the intermittency and has a fractal dimension of about 2.32. The presence of a T/NT superlayer is corroborated by the presence of a jump for the conditionally averaged streamwise velocity across the interface. The steep change in velocity is accompanied by a discontinuity in the vorticity and a jump in the Reynolds shear stress, in agreement with the governing equations within the superlayer. The analysis of the data indicates that the boundary layer entraiment is characterized by two distinctive length scales which appear to be associated with a two-stage entrainment process.

9:31AM A20.00008 Net Force Spectra of Wall Turbulence , RONALD ADRIAN, JON BALTZER, Arizona State University — In the mean momentum equation for turbulent flow the net force vector is the divergence of the turbulent Reynolds stress. In many ways it is a simpler and more fundamental quantity than the Reynolds stress. Wavenumber spectra of the streamwise net force in pipe and channel flows are shown to be similar. They define the layer structure more clearly than the mean velocity, and they show that long streamwise wavelength motions are predominantly accelerative near the wall, while short streamwise wavelengths are decelerative above the buffer layer. Spanwise wavenumber spectra provide a simpler map of the net force than the streamwise spectra, and in terms of them there is a simple boundary between accelerative and decelerative wavelengths as a function of distance above the wall.

Sunday, November 18, 2012 8:00AM - 9:44AM –

Session A21 Separated Flows I 30B - Chair: Andrzej Domaradzki, University of Southern California

8:00AM A21.00001 Numerical simulation of flow over three-dimensional dunes , UGO PIOMELLI, MOHAMMAD OMIDYEGANEH, Queen's University, Kingston (ON), Canada — We performed large-eddy simulations of the flow over a series of three-dimensional dunes at laboratory scale (the Reynolds number based on the average channel height and mean-streamwise velocity is 18,900). The three-dimensionality is imposed by shifting in the streamwise direction a standard two-dimensional dune shape according to a sine wave with an amplitude A and a wavelength λ. The three-dimensional separation of flow at the crest-line alters the distribution of pressure gradient in the spanwise direction and may result in secondary flows across the stream. The secondary flow directs low-momentum fluid, near the bed, toward the "lobe" (the most downstream point on the crest-line) and high-momentum fluid, near the free surface, toward the "saddle" (the most upstream point on the crest-line). The behaviour of the reattachment length varies depending on the induced secondary flow. Three-dimensionality increases the drag in the channel and the turbulent-kinetic energy at constant flow discharge, but the normalized TKE by the wall stress is lower than the corresponding 2D dunes. The upward flow on the stoss side and higher deceleration of flow on the lee side, over the lobe plane, elevate and broaden the separated-shear layer, respectively, affecting the TKE.

8:13AM A21.00002 Numerical simulation of the flow over Barchan dunes , MOHAMMAD OMIDYEGANEH, UGO PIOMELLI, Queen's University, Kingston (ON), Canada, KENNETH T. CHRISTENSEN, JIM BEST, University of Illinois at Urbana-Champaign — We performed large-eddy simulation of the turbulent flow over a typical barchan dune model. The configuration is similar to that of experiments carried out at the University of Illinois, but the Reynolds number based on the free-surface velocity and the dune height is one fifth of the experiment. The simulation adopts the volume-of-fluid technique to model the dune. The use of periodic boundary conditions in the streamwise and spanwise directions implies that we are considering a fully developed flow over one dune in an infinite array. The height of the domain is close to the thickness of the approaching boundary layer, upstream of the dunes in the experiment. The resolution used is close to a typical DNS; $\Delta x^+ < 20.7$, $\Delta y^+ < 0.8$, and $\Delta z^+ < 10.3$. The approaching flow to the dune accelerates over the stoss (upstream) side and rises up to the crest, while at the same time diverging slowly in the spanwise direction toward the closest horn. The separated flow either reattaches on the plane or moves helically inside the recirculation zone toward the closest horn. The separated shear-layer extends downstream and toward the free-surface and contribute to downstream dunes. The agreement of the turbulence statistics with the experiment is good.

8:26AM A21.00003 LES of a ducted propeller with rotor and stator in crashback , HYUNCHUL JANG, KRISHNAN MAHESH, University of Minnesota — A sliding interface method is developed for large eddy simulation (LES) of flow past ducted propellers with both rotor and stator. The method is developed for arbitrarily shaped unstructured elements on massively parallel computing platforms. Novel algorithms for searching sliding elements, interpolation at the sliding interface, and data structures for message passing are developed. We perform LES of flow past a ducted propeller with stator blades in the crashback mode of operation, where a marine vessel is quickly decelerated by rotating the propeller in reverse. The unsteady loads predicted by LES are in good agreement with experiments. A highly unsteady vortex ring is observed outside the duct. High pressure fluctuations are observed near the blade tips, which significantly contribute to the side-force. This work is supported by the United States Office of Naval Research.

8:39AM A21.00004 An investigation of the dynamics of marine propeller tip vortices using large-eddy simulations[1] , SETH SCHROEDER, ELIAS BALARAS, The George Washington University — The ability to capture the dynamics of tip vortices, which are generated by marine propellers, is of major interest to naval hydrodynamics designers. The tip vortex of a propeller has a direct impact on performance and acoustics. Additionally, the tip vortex is a major source of erosion damage on downstream components such as rudders and stators. In the present study we utilize large-eddy simulations to compute the flow around a generic, 7-bladed, right-handed submarine propeller in open water testing configuration. We considered three different advance coefficients at Reynolds number (based on the radius and advance speed) of the order of 300,000. The governing equations are discretized on a structured grid in cylindrical coordinates and the boundary conditions on the surface of the propeller, which is not aligned with the grid lines, are introduced using an immersed boundary method. Approximately 1 billion points is used in the computation box. Tip vortices are identified by low pressure areas and the second invariant of the velocity gradient tensor (Q-criterium). In general, the vortex core radius contracts with the acceleration in the wake, and then maintains a constant radius for a certain distance before becoming unstable. Stability is affected by the advance ratio.

[1]Work supported by ONR

8:52AM A21.00005 Large Eddy Simulation of a Gas-Turbine Model Combustor , YEE CHEE SEE, MATTHIAS IHME, University of Michigan — Gas-turbine combustors typically utilize swirling fuel-preparation strategies for flow-stabilization and flame-anchoring. Under such conditions, the flow inside the combustion chamber is highly unsteady and usually accompanied by dynamic flow structures such as precessing vortex cores. Due to this unsteadiness, steady-state flow solvers are not capable of accurately predicting the flow-field. In this study, simulations of a gas-turbine model combustor are performed using unsteady Reynolds-averaged Navier-Stokes (URANS) and large eddy simulations (LES). Simulation results are compared with experimental data to assess the capability of these modelling-techniques in predicting swirling flows under gas-turbine relevant flow-field environments.

9:05AM A21.00006 LES of Separated Flows Over an Airfoil at Moderate Reynolds Numbers[1] , G. CASTIGLIONI, J.A. DOMARADZKI, USC, M. GRILLI, S. HICKEL, TUM, Germany — Separation effects strongly affect flows for unmanned aerial vehicles, micro air vehicles, and rotating machinery, e.g., wind turbines, propellers, and low pressure turbines. The Reynolds number for such flows are usually moderate and they can be accurately simulated using DNS. However, the large computational cost of DNS makes this technique unsuitable for industrial applications while less expensive RANS and LES techniques encounter difficulties in simulating such flows because they consist of a mixture of laminar, separated, transitional, and turbulent regions. In this work we investigate an ability of LES to accurately predict the behavior of such flows on a benchmark case of a laminar separation bubble on a NACA-0012 airfoil at $Re_c = 5 \times 10^4$ at 5 deg of incidence for which accurate DNS results are available (Jones et al., JFM 602, 175 (2008)). Using a conservative immersed boundary method with the Adaptive Local Deconvolution Method (ALDM) we have performed 2D and 3D simulations of this flow with resolution reduced drastically from that in the benchmark DNS. The results to date show good predictions for the pressure coefficient C_p and the location of the separation point, but the friction coefficient C_f is not predicted accurately.

[1]Work performed during TTR40 Summer Program 2011.

9:18AM A21.00007 DNS and LES of Separated Flows at Moderate Reynolds Numbers[1] , F. CADIEUX, J.A. DOMARADZKI, USC, T. SAYADI, S. BOSE, Stanford-CTR, F. DUCHAINE, CERFACS, France — Flows in rotating machinery, for unmanned and micro aerial vehicles, wind turbines, and propellers consist of different flow regimes. First, a laminar boundary layer is followed by a laminar separation bubble with a shear layer on top of it that experiences transition to turbulence. Subsequently, the separated turbulent flow reattaches and evolves downstream from a nonequilibrium turbulent boundary layer to an equilibrium one. Typical RANS and LES turbulence modeling methods experience difficulties when simulating such flows because they were developed for fully developed turbulent flows. This currently leaves DNS as the only reliable but computationally expensive alternative. Our work assesses the capability of LES to reduce the resolution requirements for such flows. Flow over a flat plate with suitable velocity boundary conditions away from the plate to produce a separation bubble is considered. Benchmark DNS data for this configuration was generated with the resolution of 50×10^6 mesh points; also used was a different DNS database with 15×10^6 points reported by Spalart and Strelets in JFM 403 (2000). Employing two codes, one using structured and another unstructured mesh, we concluded that accurate LES are possible using O(1%) of the DNS resolution.

[1]Work performed during Stanford-CTR Summer Program 2012.

9:31AM A21.00008 Aero-optical analysis of a separated shear layer using large-eddy simulation[1] , KAN WANG, MENG WANG, University of Notre Dame — The aero-optical effects of a separated shear layer over a cylindrical turret are investigated based on the fluctuating density field obtained from large-eddy simulations. Good agreement of the velocity statistics and OPD_{rms} with experimental data validates the simulation results. The optical aberrations are compared with those caused by the turbulent boundary layer prior to separation in terms of their magnitude and structure, underlying physical mechanisms and Reynolds number dependence. It is found that the distortions by the separated shear layer are five times larger and spatially less homogeneous than those by the attached boundary layer. Pressure fluctuations are confirmed to be the main cause of aero-optical distortions, in contrast to the dominant role of entropy fluctuations in the turbulent boundary layer. With turbulent separation, the optical distortions induced by the shear layer show only weak Reynolds number dependence, which again differs from the case of a turbulent boundary layer.

[1]Supported by HEL-JTO through AFOSR Grant FA 9550-07-1-0504

Sunday, November 18, 2012 8:00AM - 9:44AM –

Session A22 Turbulence Modeling: LES/RANS 30C - Chair: Anne Staples, Virginia Polytechnic Institute and State University

8:00AM A22.00001 A posteriori analysis of spatial filters for approximate deconvolution large eddy simulations of homogeneous incompressible flows , ANNE STAPLES, OMER SAN, Virginia Tech — We investigate the effect of low-pass spatial filters for approximate deconvolution large eddy simulation (AD-LES) of turbulent incompressible flows. We propose the hyper-differential filter as a means of increasing the accuracy of the AD-LES model without increasing the computational cost. Box filters, Pade filters, and differential filters with a wide range of parameters are studied in the AD-LES framework. The AD-LES model, in conjunction with these spatial filters, is tested in the numerical simulation of the three-dimensional Taylor-Green vortex problem. The numerical results are benchmarked against direct numerical simulation (DNS) data. An under-resolved numerical simulation is also used for comparison purposes. According to the criteria used, the numerical results yield the following two conclusions: first, the AD-LES model equipped with any of these spatial filters yields accurate results at a fraction of the computational cost of DNS. Second, the most accurate results are obtained with the hyper-differential filter, followed by the differential filter.

8:13AM A22.00002 A "true" Unsteady RANS model of turbulence with inherent forcing , SUAD JAKIRLIC, ROBERT MADUTA, Darmstadt University of Technology, Institute of Fluid Mechanics and Aerodynamics, DARMSTADT UNIVERSITY OF TECHNOLOGY TEAM — Usually, a turbulence model designed and calibrated in the steady RANS (Reynolds-Averaged Navier-Stokes) framework has been straightforwardly applied to an unsteady calculation. It ended up in a steady velocity field in the case of confined wall-bounded flows; a somewhat better outcome is to be expected in globally unstable flows, such as bluff body configurations. However, only a weakly unsteady mean flow can be returned with the level of unsteadiness being by far lower compared to a referent database. Presently, an instability-sensitive, eddy-resolving model based on a differential, near-wall Reynolds stress model of turbulence is formulated and applied to several attached and separated wall-bounded configurations – channel and duct flows, external and internal flows separated from sharp-edged and continuous curved surfaces. In all cases considered the fluctuating velocity field was obtained started from the steady RANS results. The model proposed does not comprise any parameter depending explicitly on the grid spacing. An additional term in the corresponding length-scale determining equation providing a selective assessment of its production, modelled in terms of the von Karman length scale (comprising the second derivative of the velocity field) in line with the SAS (Scale-Adaptive Simulation) proposal (Menter and Egorov, 2010), represents here the key parameter.

8:26AM A22.00003 A-priori analysis of subgrid scalar flux models for turbulent high Schmidt number passive scalar mixing , SIDDHARTHA VERMA, GUILLAUME BLANQUART, California Institute of Technology — Numerical study of high Schmidt (Sc) number scalars using LES is hampered by the fact that few well-tested models exist specifically for this regime. Low Sc LES studies generally employ the widely used eddy-viscosity type models. However, these suffer from the fact that the resolved scalar gradient is often not well-aligned with the subgrid scalar flux (SGF). In this work, we perform an a-priori analysis based on Sc$\gg$1 simulations to explore the alignment of the SGF with the resolved as well as subgrid velocity and scalar quantities. Special attention is given to differences observed in the inertial-convective and the viscous-convective sub-ranges. These differences are related to the fundamental dissimilarities in the transport mechanisms. The alignment trends are used to ascertain the suitability of existing models for use in both sub-ranges.

8:39AM A22.00004 Implicit LES of turbulent flows with a high order discontinuous Galerkin method , SIMONE GREMMO, Polytechnic faculty, University of Mons, Belgium, CORENTIN CARTON DE WIART, BASTIEN GORISSEN, KOEN HILLEWAERT, Cenaero, Belgium, GREGOIRE WINCKELMANS, Universite catholique de Louvain (UCL) - iMMC, GREGORY COUSSEMENT, LAURENT BRICTEUX, Polytechnic faculty, University of Mons, Belgium — This study is concerned with the ability of a flow solver using a discontinuous Galerkin method (DGM) to perform large eddy simulation (LES) of turbulent flows. Several approaches are considered: the Smagorinsky model, a multiscale model, and implicit LES (ILES). First DGM is used to simulate 1D Burgers turbulence. This helps to analyze the spectral behavior of different subgrid scale (SGS) models. Energy spectra are obtained and compared to those obtained using a spectral code. It is shown that a moderate order (p=3,4) ILES-DGM approach provides sufficient dissipation concentrated only at the smallest resolved scales of the flow, preserving the larger scales. Second, results on homogeneous isotropic turbulence at very high Reynolds number also allow to highlight the scale selectivity of the approach. ILES-DGM provide energy spectra and energy decay results consistent with theory. At last the method is also validated on turbulent channel flow at $Re_\tau = 395, 590$. The results show a good agreement with the DNS of Moser et al. As a conclusion, for the present cases using a SGS model with DGM does not improve and can even degrade the results compared to ILES-DGM. Indeed, ILES-DGM already dissipates the small scales without influencing the larger turbulent structures.

8:52AM A22.00005 ABSTRACT WITHDRAWN –

9:05AM A22.00006 Model Invariance as a Basis for Hybrid LES-RANS Techniques , STEPHEN WOODRUFF, Computational AeroSciences Branch, NASA Langley Research Center — Advances in computing power have made hybrid Large-Eddy Simulation (LES) - Reynolds-Averaged Navier-Stokes (RANS) approaches feasible for practical problems where RANS methods alone are unsatisfactory. Hybrid methods have been most successful when RANS and LES regions can be placed so there is little interaction between them. A more effective computation would be one where LES regions are placed exactly where required by the physics. In the present work, model-invariant quantities are defined as combinations of the flow variables which yield physical results throughout the RANS/LES transition (such as the Reynolds stress, constructed from RANS and LES contributions). Model invariance provides a means for interpreting simulation results in RANS/LES transitions; the requirement that model invariance be preserved by the flow equations introduces terms that facilitate RANS/LES transitions and allow more freedom in placing those transitions. Numerical experiments are employed to demonstrate the ability of this approach to eliminate the mismatch seen between the RANS log law and the LES log law when hybrid techniques are applied to shear layers. The insensitivity of the approach to the choice of the RANS/LES blending function and other properties of the approach are also discussed

9:18AM A22.00007 Integral Methods for Validating RANS Models , DANIEL ISRAEL, Los Alamos National Laboratory — Reynolds-averaged Navier-Stokes (RANS) models typically include multiple closure coefficients which are tuned by comparison to experimental data. For self-similar flows, this process can be simplified by examining similarity solutions of the RANS equations. A further simplification is to use approximate profiles inserted into the similarity equations. However, for problems without an asymptotic self-similar state, or for which we are interested in the initial transient response, these techniques cannot be used, leaving a tedious and computationally expensive parameter search through solutions of the original multi-dimensional flow problem. Classical integral methods provide a convenient alternative approach. Substituting a guessed approximate profile into the RANS equations directly, and integrating across the layer, yields ordinary differential equations for the integral properties of the profile. These simpler equations can be used to examine the behavior of the flow in various regimes and the effects of different parameter settings.

9:31AM A22.00008 Computational and Experimental Investigations of Flow past Spinning Cylinders[1] , IGBAL MEHMEDAGIC, LIAM BUCKLEY, PASQUALE CARLUCCI, DONALD CARLUCCI, U. S. Army ARDEC, Picatinny Arsenal, NJ, ELIAS ALJALIS, SIVA THANGAM, Stevens Institute of Technology, Hoboken, NJ, STEVENS-ARDEC COLLABORATION — An anisotropic two-equation Reynolds-stress model is developed by considering the modifications to the energy spectrum and through invariance based scaling. In this approach the effect of rotation is used to modify the energy spectrum, while the influence of swirl is modeled based on scaling laws. The resulting generalized model is validated for benchmark turbulent flows with swirl and curvature. The time-averaged equations of motion and energy are solved using the modeled form of transport equations for the turbulence kinetic energy and the scalar form of turbulence dissipation with an efficient finite-volume algorithm. Computations for flow past an axially rotating cylinder with a free-spinning base are performed along with experiments for a range of spin rates and free stream flow conditions. A subsonic wind tunnel with a forward-sting mounted spinning cylinder is used for experiments. The experimental results of Carlucci & Thangam (2001) are used to benchmark flow over spinning cylinders. The data is extended to munitions spinning in the wake of other munitions and applications involving the design of projectiles are discussed.

[1]This work was funded in part by U. S. Army ARDEC.

Sunday, November 18, 2012 8:00AM - 9:44AM –

Session A23 Turbulence Theory: Isotropic Turbulence 30D - Chair: Saba Almalkie, University of Massachusetts Amherst

8:00AM A23.00001 Invariants of the reduced velocity gradient tensor in turbulent flows , JOSE CARDESA, Universidad Politecnica de Madrid, DHIREN MISTRY, LIAN GAN, JAMES DAWSON, University of Cambridge — In this paper we examine the invariants p and q of the reduced 2×2 velocity gradient tensor formed from a 2D slice of an incompressible 3D flow. Based on 2D PIV measurements and 3D DNS, we show that the joint probability density function of p and q exhibits a common characteristic shape shared across various turbulent flows. This is confirmed by data from a turbulent jet, a turbulent channel flow, isotropic turbulence in a periodic cube and mixing tank shear turbulence. The asymmetry in the shape of the resulting scatter plot is studied and proved to follow from the predominance of vortex stretching in all these flows. The only assumptions required for the proof are local homogeneity and local isotropy applied to the velocity gradients. We compare this $p - q$ scatter plot for which only 2D data is required with the widely known $Q - R$ scatter plot based on 3D information. Finally, we explore the properties of the strain rates deduced from the 2D velocity gradient tensor only. We find in which cases these can be used to unambiguously discriminate between sheet-forming or tube-forming configurations of the full 3D strain rates.

8:13AM A23.00002 Evolution of the velocity gradient invariants of fractal-generated turbulence[1] , RAFAEL FERNANDES, Imperial College London, BHARATHRAM GANAPATHISUBRAMANI, University of Southampton, CHRISTOS VASSILICOS, Imperial College London — An experimental study of turbulence generated by low-blockage space-filling fractal square grids was performed using cinematographic Stereo Particle Image Velocimetry in a water tunnel. Velocity gradient tensors were determined using Taylor's hypothesis and their invariants were computed at different distances downstream of the grid. It is shown that the classical tear-drop shape of the second and third invariant (Q and R) diagram is not seen throughout all measured stations but, instead, develops to the well known shape with downstream distance from the grid. Surprisingly, the averages of the Q and R remain zero throughout the measurements in space, even in highly inhomogeneous regions of the flow. The structure function achieves the 2/3 power law when conditioned on a very active sub-region of the flow, well before where the classical shape of the Q-R diagram is established, and in a non-Gaussian, inhomogeneous part of the turbulent flow. Finally, the alignment of the vorticity vector with the eigen vectors of the strain rate tensor in specific quadrants of the Q-R diagram is studied as a function of downstream position.

[1]EPSRC Grant EP/H030875/1

8:26AM A23.00003 The Forward-Backward Time Asymmetry in Shape Deformation of Tetrahedra in Fully Developed Turbulence , JENNIFER MUTSCHALL, HAITAO XU, Max Planck Institute for Dynamics and Self-Organization, ALAIN PUMIR, Ecole Nationale Superieure de Lyon, EBERHARD BODENSCHATZ, Max Planck Institute for Dynamics and Self-Organization — The analysis of shape deformations of multi-particle clusters can serve as an important tool for gaining insights in turbulent mixing. Recent experiments and numerical simulations on clusters of four particles (i.e. tetrahedra) observe a tendency for initially isotropic tetrahedra to deform into coplanar structures and thereby to enhance the mixing process. A quantitative understanding was to date missing. Further, the understanding of the forward-backward time asymmetry in shape deformation can elucidate the time-irreversibility of fully developed turbulence. Here we present an explanation of the observations and extend the analysis to the dynamics of tetrahedra backwards in time. We report our analytical results and compare them with our particle tracking experiments in a von Karman swirling flow and with direct numerical simulations of homogeneous isotropic turbulence in a periodic box.

8:39AM A23.00004 Intense dissipative mechanisms of strong thin shear layers in high Reynolds number turbulence , TAKASHI ISHIHARA, Nagoya University, JST CREST, JULIAN C.R. HUNT, University College London, YUKIO KANEDA, Aichi Institute of Technology — Direct numerical simulation of box turbulence at the Taylor micro-scale Reynolds number $R_\lambda = 1131$ on 4096^3 grid points was used to show that strong thin shear layers are the significant intermittent structures of high Reynolds number turbulence. Both the distance between the layers and their widths are comparable with the integral length scale L. The layers' thicknesses ℓ are of the order of the Taylor micro-scale λ. Typically $\ell \sim 4\lambda$, where $\lambda \sim 35L/R_\lambda$. Across the significant layers there are jumps in large-scale velocities of the order of the rms velocity u_o. Within the layers, much thinner intermittent, elongated vortical eddies are generated, with microscale thickness $\ell_v \sim 178L/R_\lambda^{3/2}$ with associated large peak values of vorticity of order $u_o/\ell_v (< 35\omega_{\rm rms})$ and velocities of the order of $u_o (< 3.4u_o)$, where $\omega_{\rm rms}$ is the rms vorticity. The vorticity of these micro-scale eddies have components predominantly parallel to the average vorticity of the thin shear layers. Their spacing is of order ℓ_v, so that vortices within the layers are reasonably close packed. The high relative magnitude of dissipation in the significant thin layers balances with the high relative magnitude of energy transfer (across the wave number k) for k larger than π/ℓ. The marked increase in the energy transfer inside the layer for k comparable with π/ℓ defines the eddy scales where the maximum energy transfer occurs from outside to inside.

8:52AM A23.00005 On the angle between relative velocity and relative acceleration between two fluid particles in turbulence[1] , HAITAO XU, MPI Dynamics & Self-Organization, Goettingen, Germany, ALAIN PUMIR, ENS-Lyon, Lyon, France, EBERHARD BODENSCHATZ, MPI Dynamics & Self-Organization, Goettingen, Germany — In turbulence study, it is often desirable to know if locally the flow is strain-dominated or vorticity-dominated. This information not only is related to the local flow topology, it also reveals where small particles with weak inertia accumulate. However, to determine whether strain or vorticity is dominating requires access to the velocity gradient tensor, which is difficult to measure experimentally. By using results from direct numerical simulation of fully developed turbulence we show that the angle between the relative velocity and the relative acceleration between two fluid particles can be used as an indicator of strain-dominated versus vorticity-dominated flow structure. This new indicator has the advantage that it is much more easily accessible experimentally than measuring the velocity gradients. We also present further turbulence statistics from both DNS and experiments conditioned on the angle between relative velocity and relative acceleration and compare them with those conditioned on strain and vorticity.

[1]We thank financial supports from MPG, DFG, IDRIS, and ANR.

9:05AM A23.00006 Wind tunnel measurements of scale-by-scale energy transfer, dissipation, advection and production/transport in equilibrium and nonequilibrium decaying turbulence[1] , PEDRO VALENTE, CHRISTOS VASSILICOS, Imperial College London — The cornerstone assumption that $C_\epsilon \equiv \epsilon L/u^3 \approx constant$ was found to breakdown in certain nonequilibrium regions of decaying grid-generated turbulence with wide power-law near -5/3 spectra where the behaviour of C_ϵ is, instead, very close to $C_\epsilon \sim Re_L^{-1}$ (Valente & Vassilicos, 2012 [Phys. Rev. Lett. 108, 214503]). We investigate nonequilibrium turbulence by measuring with two cross wire anemometers the downstream evolution of the scale-by-scale energy transfer, dissipation, advection, production and transport in the lee of a square-mesh grid and compare with a region of equilibrium turbulence. For the nonequilibrium case it is shown that the production and transport terms are negligible for scales smaller than about a third of L. For both cases it is shown that the peak of the scale-by-scale energy transfer scales as u^3/L which is the expected behaviour for equilibrium turbulence. However, for the nonequilibrium case this implies an imbalance between the energy transfer to the small scales and the dissipation. This imbalance is reflected on the small-scale advection which becomes larger in proportion to the maximum energy transfer as the turbulence decays whereas it stays proportionally constant in the equilibrium case.

[1]P. V. acknowledges the financial support from Fundação para a Ciência e a Tecnologia (SFRH/BD/61223/2009, cofinanced by POPH/FSE).

9:18AM A23.00007 The Reynolds number dependence of classical grid turbulence[1] , EBERHARD BODENSCHATZ, GREGORY BEWLEY, MICHAEL SINHUBER, Max Planck Institute (DS) Goettingen, MARGIT VALLIKIVI, MARCUS HULTMARK, ALEXANDER SMITS, Princeton University, INTERNATION COLLABORATION FOR TURBULENCE RESEARCH COLLABORATION — We measured inertial and dissipation range statistics in the decaying turbulence generated by a biplanar grid of crossed bars. We did so at Taylor Reynolds numbers between 130 and 1700, reaching higher than any previous study of reasonably homogeneous and isotropic turbulence. The measurements were made in the Variable Density Turbulence Tunnel at the Max Planck Institute in Göttingen with both traditional hot-wire anemometers and the new nano-fabricated NSTAP anemometers developed at Princeton. We fixed the large-scale conditions of the flow while changing the Reynolds number only by changing the viscosity of the fluid. To do this, we used two gases, air and sulfur hexafluoride, and adjusted the pressure of the gases to between 1 and 15 bar. The data confirm that even when the large-scale conditions are controlled as the Reynolds number is raised, scaling ranges are not well-defined unless Extended Self-Similarity is employed.

[1]supported by the Max Planck Society

9:31AM A23.00008 Local and distant interactions in the Batchelor regime of scalar turbulence , ROBERT RUBINSTEIN, None, WOUTER BOS, LMFA-CNRS Universite de Lyon, Ecole Centrale de Lyon FRANCE — Kraichnan's 1968 paper on the passive scalar reconsidered Batchelor's classic analysis of the persistence of scalar fluctuations in the dissipation range of the velocity field when the scalar diffusivity is much smaller than the fluid viscosity. Adopting the premise that the velocity field fluctuates rapidly, instead of Batchelor's hypothesis that the velocity field is essentially static, Kraichnan found that although Batchelor's prediction of a k^{-1} spectrum remains intact, the subsequent diffusive range falls off as $\exp(-k)$ instead of Batchelor's prediction $\exp(-k^2)$. We will show that these two hypotheses also make significantly different predictions of higher order statistics in the k^{-1} range, namely that in Kraichnan's analysis, a reduction of mean square advection analogous to the 'suppression of nonlinearity' in Navier-Stokes turbulence occurs, but that this effect is absent in Batchelor's analysis. This difference will be interpreted in the light of a suggestion of Yukio Kaneda that the difference between Kraichnan's and Batchelor's analysis originates in treating velocity-scalar interactions either as local (Kraichnan) or as asymptotically distant (Batchelor).

Sunday, November 18, 2012 8:00AM - 9:44AM –

Session A24 Acoustics I: Turbulence and Aerodynamics 30E - Chair: Daniel Bodony, University of Illinois at Urbana-Champaign

8:00AM A24.00001 Discrete tones around airfoils: a global stability analysis.[1] , MIGUEL FOSAS DE PANDO, PETER J. SCHMID, LadHyX, CNRS-Ecole Polytechnique, DENIS SIPP, ONERA - DAFE — Airfoil self-noise stems from an interaction between the airfoil surface, the boundary layers and the wake. At moderate Reynolds number and small angles of attack, the acoustic spectrum is dominated by discrete tones correlated to the ringing of coherent structures localized in the vicinity of the trailing edge. Local stability analyses show strong amplification of hydrodynamic instabilities in the frequency range of acoustic tones, suggesting an interplay between sound waves and instabilities. However, owing to the intrinsic limitations of local approaches, a satisfactory explanation of the tonal-noise phenomenon is still missing. We present a global stability analysis of the mean-flow linearized dynamics. Features of the global modes spectrum and of the resolvent norm will be discussed. The least-stable direct modes show a link between the suction-side boundary layer, the near wake dynamics, and acoustic radiation; conversely, the corresponding adjoint modes pinpoint at the pressure side as the location of maximum sensitivity. Although the linearized operator is stable, the resolvent norm shows substantial energy amplification. Finally, the pressure-side, suction-side and wake dynamics will be analyzed in isolation to assess their respective contribution to the overall process.

[1]This work was performed using HPC resources from GENCI-CINES (Grant 2012-026451)

8:13AM A24.00002 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. Details of the development including arbitrary surface extraction techniques on unstructured grids and a novel end-cap correction for the quadrupole sound will be presented. 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. We have used these techniques to compute the noise from high Reynolds number cylinder flows and compare our results against available computations and experiments; base flow results as well as acoustic data will be compared.

[1]Office of Naval Research

8:26AM A24.00003 Numerical investigation of acoustic radiation from vortex-airfoil interaction[1] , ANNE LEGAULT, MINSUK JI, MENG WANG, University of Notre Dame — Numerical simulations of vortices interacting with a NACA 0012 airfoil and a flat-plate airfoil at zero angle of attack are carried out to assess the applicability and accuracy of classical theories. Unsteady lift and sound are computed and compared with the predictions by theories of Sears and Amiet, which assume a thin-plate airfoil in an inviscid flow. A Navier-Stokes solver is used in the simulations, and therefore viscous effects are taken into consideration. For the thin-plate airfoil, the effect of viscosity is negligible. For a NACA 0012 airfoil, the viscous contribution to the unsteady lift and sound mainly comes from coherent vortex shedding in the wake of the airfoil and the interaction of the incoming vortices with the airfoil wake, which become stronger at higher Reynolds numbers for a 2-D laminar flow. When the flow is turbulent at chord Reynolds number of 4.8×10^5, however, the viscous contribution becomes negligible as coherent vortex shedding is not present. Sound radiation from vortex-airfoil interaction at turbulent Reynolds numbers is computed numerically via Lighthill's theory and the result is compared with the predictions of Amiet and Curle. The effect of the airfoil thickness is also examined.

[1]Supported by ONR Grant N00014-09-1-1088

8:39AM A24.00004 Effects of step non-compactness and free-stream convection on step noise[1] , JIN HAO, AHMED ELTAWEEL, MENG WANG, University of Notre Dame — The effects of acoustic non-compactness of steps and free-stream convection on sound generation in a turbulent boundary layer over small steps are investigated in the Lighthill framework by comparing solutions obtained using a boundary-element method with those based on a compact-step Green's function. When the ratio of acoustic wavelength to the step height, λ/h, is large, good agreement between the two is found in terms of sound spectra and directivity. Discrepancies become significant with decreasing λ/h. For mildly non-compact step heights, the sound directivity for forward steps exhibits asymmetry, with the upstream side maintaining an approximate dipole lobe while the rest significantly distorted. Dips in sound spectra and multiple lobes in directivity are observed along with significantly enhanced sound power when λ/h is sufficiently small, indicating that edge-scattering becomes the dominant source mechanism. The effect of convection on the far-field sound is found to be significant at free-stream Mach number of 0.1 or higher. These results indicate that both step non-compactness and convection effects must be taken into consideration when performing wind-tunnel experiments for hydroacoustic applications.

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

8:52AM A24.00005 Low Mach number prediction of the acoustic signature of fractal-generated turbulence , SYLVAIN LAIZET, Imperial College London, VÉRONIQUE FORTUNÉ, ERIC LAMBALLAIS, Institut P', CHRISTOS VASSILICOS, Imperial College London — We compare the acoustic properties of a fractal square grid with those of a regular grid by means of a hybrid approach based on Lighthill's analogy and Direct Numerical Simulation (DNS). Our results show that the sound levels corresponding to our fractal square grid of three fractal iterations are significantly reduced by comparison to a regular grid of same porosity and mesh-based Reynolds number. We also find a well-defined peak at a Strouhal number between 0.2 and 0.3 in the acoustic spectrum of our fractal square grid which is absent in the case of our regular grid. We explain this effect in terms of a new criterion for quasi-periodic vortex shedding from a regular or fractal grid.

9:05AM A24.00006 Mechanisms of "crackle" acoustic radiation from high speed turbulence , AARON ANDERSON, JONATHAN FREUND, University of Illinois at Urbana-Champaign — "Crackle" describes the perception of a particular type of jet noise produced by high specific thrust engines, which is both annoying and potentially damaging to hearing. Direct numerical simulations of free shear flows at Mach numbers ranging from $M = 1.5$ to 3.5 are used to investigate its source mechanisms. Shear layers with $M > 2$ are seen to produced radiation with the accepted character of crackling jets: weak shocks, asymmetric and apparently steepened pressure waves, sound pressure levels exceeding 160 dB, and a distinctive Mach angle in the near acoustic field. Space–time correlations suggest that eddy advection is indeed responsible for the Mach-wave radiation. However, the length and time scales of the space–time statistics are significantly smaller than would be expected based upon the observed Mach waves, which suggests that near-field agglomeration of the waves is an essential feature. The pressure skewness, a metric that correlates with perception of crackle, is near zero for $M = 1.5$ and increases above 0.6 for $M = 3.5$ in the near acoustic field. Within the simulation domain, skewness is shown to increase with increasing distance from the turbulent shear layer indicative of multi-dimensional wave interaction effects.

9:18AM A24.00007 Modeling Unsteady Lift and Radiated Sound Generated by a 2-D Airfoil in an Intermittent Flow , MARK ROSS, SCOTT MORRIS, University of Notre Dame — The spanwise correlation length scale of lateral velocity and the gust response function are the quantities of interest in predicting the sound production from an airfoil. Typically, these quantities are taken to be a correlation length scale model based on isotropic turbulence and Sears' gust response function, respectively. The present study is an experimental investigation of the accuracy of these selections when the turbulent approach flow is intermittently irrotational. Acoustic measurements of a flat-plate airfoil placed at three lateral locations in a single stream shear layer are presented. The acoustic measurements are compared to radiated sound predictions based on detailed velocity field measurements. A potential model which accounts for the effect of approach flow intermittency on the radiated sound will also be presented.

9:31AM A24.00008 Poroelastic Trailing Edge Noise and the Silent Flight of the Owl , JUSTIN JAWORSKI, NIGEL PEAKE, University of Cambridge — Many species of owl rely on specialised plummage to reduce their self-noise levels and enable hunting in acoustic stealth. One such plummage arrangement, a compliant array of feathers at the wing trailing edge, is believed to mitigate the scattering of boundary layer turbulence which is the predominant source of airframe noise. The owl noise problem is modelled analytically by the diffraction of a quadrupole source by a semi-infinite porous and elastic edge, and the resulting set of equations is solved exactly using the Wiener-Hopf technique to identify important dimensionless parameters and their scaling behaviour with respect to the aerodynamic noise produced. Special attention is paid to the limiting cases of elastic-impermeable as well as rigid-porous plate conditions, the latter of which is compared against available experimental measurements in the literature. Results from this analysis and comparison seek to validate the weaker sixth-power dependence of far-field acoustic power on flow velocity for porous trailing edges, develop a rigorous basis for the aeroacoustic tailoring of poroelastic edges to reduce airframe noise, and help explain one of the mechanisms of aerodynamic noise suppression by owls.

Sunday, November 18, 2012 8:00AM - 9:44AM –

Session A25 Flow Control: General I 31A - Chair: Jesse Little, University of Arizona

8:00AM A25.00001 Maximum-entropy principle as Galerkin modelling paradigm[1] , BERND R. NOACK, Institute PPRIME, France, ROBERT K. NIVEN, ADFA/UNSW, Australia, CLARENCE W. ROWLEY, Princeton University, USA — We show how the empirical Galerkin method, leading e.g. to POD models, can be derived from maximum-entropy principles building on Noack & Niven 2012 JFM. In particular, principles are proposed (1) for the Galerkin expansion, (2) for the Galerkin system identification, and (3) for the probability distribution of the attractor. Examples will illustrate the advantages of the entropic modelling paradigm.

[1]Partially supported by the ANR Chair of Excellence TUCOROM and an ADFA/UNSW Visiting Fellowship

8:13AM A25.00002 Optimal mode decomposition for unsteady and turbulent flows , ANDREW WYNN, DAVID PEARSON, Imperial College London, UK, BHARATHRAM GANAPATHISUBRAMANI, University of Southampton, UK, PAUL GOULART, ETH, Zurich — A new method, which we refer to as Optimal Mode Decomposition (OMD), to identify a linear model for the evolution of a fluid flow is presented. The method enables an ensemble of snapshot data to be used to estimate the linear dynamics of a flow by identifying a low order subspace of the flow and constructing dynamics on that low order subspace. An iterative procedure is used to find the optimal combination of linear model and subspace that minimises the system residual error. The OMD method is shown to be a generalisation of Dynamic Mode Decomposition (DMD), in which the subspace is not optimised but rather fixed to be the one spanned by the POD modes. A comparison between OMD and DMD is made using both a synthetic waveform and an experimental data set. The OMD technique is shown to have lower residual errors than DMD and is shown on a synthetic waveform to provide more accurate estimates of the system eigenvalues. This new method can be used with experimental and numerical data to calculate the 'optimal' low-order model with a user-defined rank that best captures the system dynamics of unsteady and turbulent flows.

8:26AM A25.00003 The Influence of Relative Humidity on Dielectric Barrier Discharge Plasma Flow Control Actuator Performance[1] , M. WICKS, F.O. THOMAS, T.C. CORKE, University of Notre Dame, M. PATEL, Innovative Technology Applications Company, LLC. — Dielectric barrier discharge (DBD) plasma actuators possess numerous advantages for flow control applications and have been the focus of several previous studies. Most work has been performed in relatively pristine laboratory settings. In actual flow control applications, however, it is essential to assess the impact of various environmental influences on actuator performance. As a first effort toward assessing a broad range of environmental effects on DBD actuator performance, the influence of relative humidity (RH) is considered. Actuator performance is quantified by force balance measurements of reactive thrust while RH is systematically varied via an ultrasonic humidifier. The DBD plasma actuator assembly, force balance, and ultrasonic humidifier are all contained inside a large, closed test chamber instrumented with RH and temperature sensors in order to accurately estimate the average RH at the actuator. Measurements of DBD actuator thrust as a function of RH for several different applied voltage regimes and dielectric materials and thicknesses are presented. Based on these results, several important design recommendations are made.

[1]This work was supported by Innovative Technology Applications Company (ITAC), LLC under a Small Business Innovation Research (SBIR) Phase II Contract No. N00014-11-C-0267 issued by the U.S. Department of the Navy.

8:39AM A25.00004 Turbine blade cooling using Coulomb repulsion , ROBERT BREIDENTHAL, University of Washington, JOSEPH COLANNINO, JOHN DEES, DAVID GOODSON, IGOR KRICHTAFOVITCH, TRACY PREVO, ClearSign Combustion Corp. — Video photography and thermocouples reveal the effect of an electric field on the flow around a stationary, idealized turbine blade downstream of a combustor. The hot products of combustion naturally include positive ions. When the blade is an electrode and elevated to a positive potential, it tends to attract the free electrons and repel the positive ions. Due to their lower mass, the light electrons are rapidly swept toward the blade, while the positive ions are repelled. As they collide with the neutrals in the hot gas, the positive ions transfer their momentum so that a Coulomb body force is exerted on the hot gas. Cool, compressed air is injected out of the stationary blade near its leading edge to form a layer of film cooling. In contrast to the hot combustion products, the cool air is not ionized. At the interface between the hot gas and the cool air, the Coulomb repulsion force acts on the former but not the latter, analogous to gravity at a stratified interface. An effective Richardson number representing the ratio of potential to kinetic energy characterizes the topography of the interface. When the electric field is turned on, the repulsion of the hot gas from the idealized blade is evident in video recordings and thermocouple measurements.

8:52AM A25.00005 On Optimal Model Identification in Hydrodynamics , BARTOSZ PROTAS, McMaster University, VLADISLAV BUKSHTYNOV, Stanford University, BERND NOACK, Institut PPRIME, CNRS – Universite de Poitiers, MAREK MORZYNSKI, Poznan University of Technology — This work is motivated by two classes of problems, namely, identification of temperature–dependent material properties in complex thermo–fluid phenomena and identification of inertial manifolds in reduced–order models of hydrodynamic instabilities. It is demonstrated that these two problems can be framed in terms of the reconstruction of constitutive relations and we propose a robust computational approach to solve such problems using an optimization formulation based on some measurements. A special property of this formulation is that the control variable is a function of the state (i.e., the dependent variable), rather than the independent variable, and the main novelty is that the constitutive relation is determined in a very general form with no a priori assumptions other than smoothness. The optimization problem is solved using a gradient–descent method in which the cost functional gradients exhibit structure quite different than in typical optimization problems for differential equations. As an application, the proposed identification technique will be used to determine corrections to well–known models such as the Landau equation and the mean–field model so that they capture more accurately the behavior of actual hydrodynamic systems.

9:05AM A25.00006 Global Model Reduction for Fluid-Structure System[1] , MINGJUN WEI, MIN XU, TAO YANG, New Mexico State University — There are many challenges in the numerical simulation of a problem involving fluid flow and moving solid structures, especially when fully-coupled motion is considered. The challenge becomes even greater when a reduced-order model is required for the purposes of control and optimization of such complex and coupled systems. Here, we first introduce a global formulation of fluid and solid in a uniform Eulerian framework, which works for both prescribed and coupled moving structures in fluid flow. Based on the same formulation, we propose then to have a global model reduction by applying POD/Galerkin projection on a uniform Eulerian description of fluid, structure and their interaction. Preliminary results are shown as the approach being applied to the cases with either prescribed or coupled motion.

[1]Supported by ARL

9:18AM A25.00007 Mixing Layer Excitation by Dielectric Barrier Discharge Plasma Actuators[1] , RICHARD ELY, JESSE LITTLE, University of Arizona — The response of a mixing layer with velocity ratio 0.28 to perturbations near the high-speed side (U_2=11 m/s, $Re_L = 0.26 \times 10^6$) of its origin from dielectric barrier discharge plasma actuators is studied experimentally. Both alternating current (ac) and nanosecond (ns) pulse driven plasma are investigated in an effort to clarify the mechanisms associated with each technique as well as the more general physics associated with flow control via momentum-based versus thermal actuation. Ac-DBD plasma actuators, which function through electrohydrodynamic effects, are found to generate an increase in mixing layer momentum thickness that is strongly dependent on forcing frequency. Results are qualitatively similar to previous archival literature on the topic employing oscillating flaps. Ns-DBD plasma, which is believed to function through thermal effects, has no measureable influence on the mixing layer profile at similar forcing conditions. In the context of previous archival literature, these results suggest different physical mechanisms govern active control via ac- and ns-DBD plasma actuation and more generally, momentum versus thermal perturbations. Further investigation of these phenomena will be provided through variation of the boundary/mixing layer properties and forcing parameters in the context of spatially and temporally resolved experimental data.

[1]Supported by: AFOSR and Raytheon Missile Systems

9:31AM A25.00008 Passive control and sensitivity analysis of thermo-acoustic systems via adjoint equations[1] , LUCA MAGRI, MATTHEW JUNIPER, University of Cambridge — We take a technique developed for the analysis of hydrodynamic stability and adapt it to thermo-acoustic systems. We aim to determine how thermo-acoustic systems should be changed in order to extend their linearly stable region. This technique uses adjoint equations to calculate the system's sensitivity to feedback mechanisms and to changes in the base state. We investigate two thermo-acoustic systems: a Rijke tube 1) electrically heated by a hot wire and 2) heated by a compact diffusion flame. The calculation of the components of the structural sensitivity tensor reveals the passive control mechanism that has the strongest influence on both the growth rate and frequency of thermo-acoustic oscillations. We illustrate the base-state sensitivity by calculating the effects of tiny variations of the base-state parameters. The successful application of adjoint sensitivity analysis to thermo-acoustics opens up new possibilities for the passive control of thermo-acoustic oscillations by providing gradient information that can be combined with constrained optimization algorithms in order to reduce linear growth rates.

[1]This work was supported by the European Research Council through Project ALORS 2590620.

Sunday, November 18, 2012 8:00AM - 9:44AM –

Session A26 Reactive Flows I: Turbulent Combustion Experiments 31B - Chair: Tim Lieuwen, Georgia Institute of Technology

8:00AM A26.00001 Dynamics of a Variable Density Ratio, Reacting Jet Issuing into a Vitiated Crossflow , BENJAMIN WILDE, JERRY SEITZMAN, TIM LIEUWEN, Georgia Institute of Technology — This work reports recent experimental characterization of a reacting jet issuing into a turbulent vitiated crossflow. Previous studies on unforced, non-reacting jets in crossflow showed that the flow transitions from global to convective instability with increasing jet-to-crossflow momentum flux ratio, J, whose value is a function of the jet-to-crossflow density ratio, ρ_j/ρ_∞. This work utilizes a new facility designed to study both of these stability boundaries in a reacting configuration, where the densities of the inflow jet, approaching crossflow, and flame can be systematically varied. The jet, consisting of varying mixtures of CH_4, H_2, N_2, and He, enters the test section through a flush-mounted contoured nozzle. The density ratio spans from 0.4 to 1.0 as a function of the jet constituent concentrations. The vitiated crossflow temperature, T_∞, varies from 1000 to 1600K, and the J range is 2 to 25. High-frame-rate imaging and PIV measurements show evidence of narrowband, self-excited fluctuations. Cross correlations computed from the windward and leeward flame edge motion show a tendency towards classical wake-like sinuous motion at lower J and jet-like varicose motion at higher J. Varying crossflow temperature alters the flame stabilization location but not the spectral content of the flow.

8:13AM A26.00002 Simultaneous krypton PLIF, LII and PIV measurements in a sooting non-premixed jet flame , OLIVER BUXTON, ROSS BURNS, NOEL CLEMENS, The University of Texas at Austin — Simultaneous krypton planar laser induced fluorescence (PLIF), laser induced incandescence (LII) and stereoscopic PIV measurements are made in a sooting jet flame for the first time. Krypton is seeded into the jet stream to provide a conserved scalar marker from which mixture fraction can be inferred. 2% krypton is seeded into a non-premixed ethylene/nitrogen jet flame with a cold jet Reynolds number based on bulk velocity of 5100. An electronic transition is accessed through a two photon excitation using an incident 214.7 nm light sheet, producing fluorescence at 760 nm. This fluorescence is spectrally isolated from the flame's bulk luminescence and PAH fluorescence. The fluorescence signal can then be converted to mixture fraction by measuring the relative krypton quenching rates of the various species present in the flame core and making use of a flamelet-based chemical state relationship. This technique is used simultaneously with LII, which will provide the soot volume fraction, and stereoscopic PIV which will provide all three velocity components in a plane. It will thus be possible to observe the influence of the velocity field on the formation and transport of soot, and the evolution of mixture fraction, and its effect on soot formation, within the sooting jet flame.

8:26AM A26.00003 Experimental investigation of self-turbulent flames , CHRISTOPHE ALMARCHA, Aix-Marseille University, JOEL QUINARD, CNRS — When propagating downwards, premixed flames undergo hydrodynamic instabilities. The resulting dynamics exhibits multiple corrugations of the light emitting reaction zone. By changing the reactive mixture composition or the shape of the propagation volume, the characteristic lengths of perturbation are changed. We present here the experimental study of propane-air and methane-air flames propagating in vertical circular tubes and in vertically oriented Hele-Shaw cells. This last configuration allows comparison with two dimensional numerical models. The thermo acoustic instability, usually acting when flames propagate in confined volume, is damped thanks to an acoustic absorber, allowing the study of wide flames at the meter scale.

8:39AM A26.00004 Turbulent Premixed Combustion in V-flames: Statistics of Flame Front Position , SINA KHEIRKHAH, ÖMER GÜLDER, University of Toronto Institute for Aerospace Studies — Flame front characteristics of turbulent premixed V-flames were experimentally investigated using Mie scattering and particle-image-velocimetry techniques. Experiments were performed at three mean bulk flow velocities of 4, 6.2, and 8.6 m/s along with three fuel-air equivalence ratios of 0.7, 0.8, and 0.9. Effects of the vertical distance from the flame-holder, mean bulk flow velocity, and fuel-air equivalence ratio on statistics of the flame front position were studied. Results show that, mean and RMS of distance between the flame front and the vertical axis increase with increasing the vertical distance from the flame-holder. At a fixed vertical distance above the flame-holder, mean and RMS of the distance between the flame front and the vertical axis decrease with increasing the mean bulk flow velocity; however, these statistics increase with increasing the fuel-air equivalence ratio. Results show that probability-density-function of the distance between the flame front and the vertical axis features a bell-shaped distribution. Power spectral analysis of the flame front position shows that, for all experimental conditions tested, the averaged and normalized power-spectrum-densities of the flame front position collapse and show a power-law relation with the wave number.

8:52AM A26.00005 Vortex Breakdown in a Swirl-Stabilized Combustor , ZVI RUSAK, RPI, COU UMEH, Nestoil PLC, EPHRAIM GUTMARK, UC — Results of lean premixed reacting flow tests in a swirl-stabilized combustor show the complex interaction between the flame and the vortex breakdown (VB) zone and oscillations in the position of both. PIV measurements give the detailed velocity field, from which the swirl ratio for a given swirler is computed. Position and size of the VB zone are also determined at various equivalence ratios. Simultaneous OH chemiluminescence snapshots identify the location of the flame. For the given setup with a fixed swirler, upstream pressure and mass flux, it is found that the VB zone occurs near the expansion plane in the nonreacting cold flow case and is pushed downstream when flow is preheated. For low equivalence ratio (Φ) near the flammability limit (0.48 to 0.52), the VB is anchored at the expansion plane and the flame oscillates inside it. At higher Φ (0.55 to 0.65), the VB zone apex and flame front are close to each other and oscillate together near the expansion plane. At even higher Φ (> 0.7), the flame is anchored at the expansion corners while the VB zone oscillates downstream of it. A theoretical discussion that is based on the compressible flow vorticity transport equation sheds light on the various mechanisms that govern VB position in reacting flows.

9:05AM A26.00006 High speed OH-PLIF measurement of self-excited circumferential instabilities in an annular combustion chamber , NICHOLAS WORTH, JAMES DAWSON, University of Cambridge — Self-excited thermo-acoustic instabilities are a significant issue in the development of lean burn gas turbine combustors. Such instabilities arise through coupling of the unsteady heat release and acoustic waves, which can propagate both longitudinally and circumferentially in annular combustor geometries. Although a large number of studies have investigated longitudinal fluctuations in single axisymmetric flames, it is currently uncertain whether these results can be used to emulate circumferential oscillations in annular geometry. Therefore, the aim of the current project is to investigate the flame dynamics in an annular model gas turbine combustor during self-excited circumferential oscillations. Pressure measurements are used to characterise the circumferential oscillations, with high-speed OH chemiluminescence and OH-PLIF used to capture the flame dynamics. The flame structure and dynamics are significantly affected by both the proximity of neighbouring flames and the excitation mode; with different responses observed for small and large separation distances, and standing and spinning modes. These observations indicate that results from single flame investigations may only be representative of self-excited flames in annular geometry under in a limited set of conditions.

9:18AM A26.00007 Turbulent Flame Speed Scaling for Positive Markstein Number Expanding Flames in Near Isotropic Turbulence[1] , SWETAPROVO CHAUDHURI, FUJIA WU, CHUNG LAW, Princeton University — In this work we clarify the role of Markstein diffusivity on turbulent flame speed and it's scaling, from analysis and experimental measurements on constant-pressure expanding flames propagating in near isotropic turbulence. For all C0-C4 hydrocarbon-air mixtures presented in this work and recently published C8 data from Leeds, the normalized turbulent flame speed data of individual mixtures approximately follows the recent theoretical and experimental $Re_{T,f}^{0.5}$ scaling, where the average radius is the length scale and thermal diffusivity is the transport property. We observe that for a constant $Re_{T,f}^{0.5}$, the normalized turbulent flame speed decreases with increasing Mk. This could be explained by considering Markstein diffusivity as the large wavenumber, flame surface fluctuation dissipation mechanism. As originally suggested by the theory, replacing thermal diffusivity with Markstein diffusivity in the turbulence Reynolds number definition above, the present and Leeds dataset could be scaled by the new $Re_{T,f}^{0.5}$ irrespective of the fuel considered, equivalence ratio, pressure and turbulence intensity for positive Mk flames.

[1]This work was supported by the Combustion Energy Frontier Research Center funded by the U.S. Department of Energy, Office of Basic Energy Sciences under Award Number DE-SC0001198 and by the Air Force Office of Scientific Research.

9:31AM A26.00008 Oscillatory Flame Response in Acoustically Driven Fuel Droplet Combustion[1] , CRISTHIAN SEVILLA, AYABOE EDOH, JEFFREY WEGENER, AARON SUNG, KELVIN CHEN, BRETT LOPEZ, OWEN SMITH, ANN KARAGOZIAN, University of California, Los Angeles — This experimental study focuses on combustion of liquid fuel droplets during exposure to external acoustic disturbances generated as standing waves within a closed acoustic waveguide. Both visible imaging as well as phase-locked OH* chemiluminescence imaging are used to quantify flame motion and response during such excitation. Acoustic perturbations create both a mean flame deflection as well as flame front oscillations in time that can be dependent on the droplet's location relative to the pressure node (PN) or pressure antinode (PAN) in the waveguide. A range of acoustic forcing frequencies and droplet locations can be used to investigate flame movement. Phase-locked OH* chemiluminescence imaging reveals not only a deflected flame which oscillates in position relative to the droplet, but also different degrees of oscillation depending on excitation frequency and droplet position within the waveguide; there are also oscillations in localized flame front chemiluminescent intensity. Results for differences in flame dynamics are explored in the context of the well-known Rayleigh criterion, with implications for other non-premixed reactive systems.

[1]Supported by the US Air Force/ERC Inc.

Sunday, November 18, 2012 8:00AM - 9:44AM –

Session A27 Rarefied Gases 31C - Chair: John Sader, California Institute of Technology

8:00AM A27.00001 The Effect of Rotational Non-equilibrium on Chemical Reaction Rates Predicted by the Quantum-Kinetic (Q-K) Model for Direct Simulation Monte Carlo (DSMC) Simulations , M.A. GALLIS, J.R. TORCZYNSKI, Sandia National Laboratories — The effect of non-equilibrium rotational excitation on dissociation and exchange reaction rates predicted by Bird's Q-K model is analyzed. The effect of rotational non-equilibrium is introduced as a perturbation to the effect of vibrational non-equilibrium. For dissociation reactions, a small but measurable improvement in the rates is observed. For exchange reactions, the change is negligible. These findings agree with experimental observations and theoretical predictions. The results from one-dimensional stagnation-streamline and two-dimensional axi-symmetric DSMC implementations of the original and modified Q-K models are compared for a typical re-entry flow. The influence of rotational non-equilibrium in promoting chemical reactions is seen to be small for this type of flow. Sandia National Laboratories is a multi-program laboratory managed and operated by Sandia Corporation, a wholly owned subsidiary of Lockheed Martin Corporation, for the U.S. Department of Energy's National Nuclear Security Administration under contract DE-AC04-94AL85000.

8:13AM A27.00002 Asymptotic analysis of the Boltzmann-BGK equation for oscillatory gas flows with application to thermal creep[1] , JASON NASSIOS, JOHN SADER[2], The University of Melbourne, Department of Mathematics and Statistics — Kinetic theory provides a rigorous foundation for calculating the dynamics of gas flow at arbitrary degrees of rarefaction. Solutions to the Boltzmann equation require numerical methods in many cases of practical interest. However, the near-continuum regime has been analyzed analytically using asymptotic techniques. These asymptotic analyses often assume steady flow, for which analytical slip models have been derived. Recent developments in nanofabrication have stimulated research into the study of oscillatory flows, drawing into question the applicability of the steady flow assumption. In this talk, I will discuss some key findings of a formal asymptotic analysis of the unsteady linearized Boltzmann-BGK equation, which generalizes existing theory to the unsteady case. The near-continuum limit is considered where the mean free path and oscillation frequency are small. A brief exploration of the implications of this theory for the oscillatory thermal creep problem will be presented, where temperature gradients along adjacent walls generate a flow.

[1]The authors acknowledge financial support from an Australian Postgraduate Award and the Australian Research Council Grants Scheme.
[2]Currently at Kavli Nanoscience Institute, California Institute of Technology, Pasadena, California 91125

8:26AM A27.00003 Numerical study of oscillatory Couette flow in rarefied gas , YING WAN YAP, JOHN SADER[1], Department of Mathematics and Statistics, The University of Melbourne, Victoria 3010, Australia, YONG SHI, School of Power Engineering, Chongqing University, Chongqing 400030, People's Republic of China — Gas flows generated by nanoscale devices can achieve oscillation frequencies comparable to the intermolecular collision frequency. Modeling these flows often requires the use of kinetic theory, because such operation invalidates the use of standard continuum treatments. The Bhatnagar-Gross-Krook (BGK) kinetic model approximates the effect of molecular collisions as a relaxation process, and is often used to describe non-equilibrium gas flows. In this talk, I will present a numerical study of oscillatory Couette flow to obtain benchmark solutions using the BGK model. Applicability of the lattice Boltzmann method to such flow is also assessed using these benchmark solutions.

[1]Kavli Nanoscience Institute and Department of Physics, California Institute of Technology, Pasadena, California, 91125, USA

8:39AM A27.00004 Modeling fluid flows in micro devices: the challenge of Knudsen-layer behavior , NISHANTH DONGARI, YONGHAO ZHANG, JASON REESE, Department of Mechanical and Aerospace Engineering, University of Strathclyde, Glasgow G1 1XJ, UK, MULTISCALE FLOWS RESEARCH GROUP TEAM — Microscale gas flows often display non-standard fluid behavior and near a solid bounding surface. This leads to the formation of a Knudsen layer (KL): a local non-equilibrium region of thickness of few mean free paths (MFP) from the surface. Linear constitutive relations for shear stress and heat flux are no longer necessarily valid in the KL. To account this, we investigate a power-law (PL) form of the probability distribution function for free paths of rarefied gas molecules in arbitrary wall confinements. PL based geometry dependent MFP models are derived for complex geometry systems by taking into account the boundary limiting effects on the molecular free paths. Molecular dynamics numerical experiments are carried out to rigorously validate the PL model, under wide range of rarefaction conditions. As gas transport properties can be related to the MFP through kinetic theory, the Navier-Stokes-Fourier (N-S-F) constitutive relations are then modified in order to better capture the flow behavior in the KL. The new modeling technique tested for isothermal and non-isothermal gas flows in both planar and non-planar confinements. The results show that our approach greatly improves the near-wall accuracy of the N-S-F equations, well beyond the slip-flow regime.

8:52AM A27.00005 Exploiting coupled heat and momentum transfer in nanostructured gas-filled channels , TOBIAS BAIER, STEFFEN HARDT, TU Darmstadt, Center of Smart Interfaces — The velocity distribution of a gas confined between surfaces held at different temperatures shows a significant deviation from the Maxwell distribution as long as the mean free path of the molecules is comparable to the channel dimensions. When one of the surfaces is suitably structured this non-equilibrium distribution can be exploited to transfer momentum in tangential direction between the two surfaces. Hence the walls experience a net force parallel to their surface which opens the possibility to extract work from the system. Since both surfaces are held at constant temperature this mode of momentum transfer is different from thermal creep flow that has gained more attention so far. We investigate this situation in the limit of free molecular flow for the case that the unstructured surface is allowed to move tangentially with respect to the structured surface. By this the possibility to operate the system as a thermal engine is investigated and its efficiency assessed.

9:05AM A27.00006 Flow of a rarefied gas around moving vanes in Crookes radiometer: Numerical analysis of a model problem , SATOSHI TAGUCHI, Department of Mechanical Engineering and Intelligent Systems, The University of Electro-Communications, KAZUO AOKI, Department of Mechanical Engineering and Science, Kyoto University — A model for flows around moving vanes in a Crookes radiometer is proposed. More precisely, a series of uniformly spaced parallel flat plates heated on their single sides, moving in a channel at a constant speed, is considered in the case where the direction of motion is perpendicular to the plates. The flow around the plates is investigated numerically on the basis of the ellipsoidal statistical (ES) model of the Boltzmann equation and the diffuse reflection boundary condition by means of an accurate finite-difference method. Special attention is paid to the discontinuity contained in the velocity distribution function. The pressure distribution around the edge is found to be different from that reported previously [S. Taguchi and K. Aoki, J. Fluid Mech. **694**, 191–224 (2012)], in which the plate is assumed to have a stationary position in a closed container. The reason for the difference is also discussed.

9:18AM A27.00007 Thermal transpiration of a slightly rarefied gas through a horizontal straight pipe in the presence of weak gravitation , TOSHIYUKI DOI, Tottori University — Thermal transpiration of a slightly rarefied gas in the presence of weak gravitation is studied based on kinetic theory. We consider the situation where the Knudsen number (the mean free path divided by the characteristic length of the cross section) is small and the dimensionless gravity (the characteristic length divided by the ascent height of the molecules against gravity) is of the order of the square of the Knudsen number. The behavior of the gas is studied analytically based on the the fluid-dynamic-type equation derived from the Boltzmann equation. When the temperature gradient is imposed along the pipe, the pressure gradient is produced not only in the vertical direction but also in the horizontal direction due to the effect of gravity. This horizontal pressure gradient, which is of the higher order of the Knudsen number, induces a flow of the order of the Knudsen number, and produces a relatively finite effect on thermal transpiration. The velocity profile is considerably different from that of the conventional thermal transpiration due to the effect of weak gravitation. A direct numerical analysis of a flow through a long channel is conducted based on the model Boltzmann equation, and the mechanism of this phenomenon is demonstrated.

9:31AM A27.00008 Lattice Boltzmann simulations of genuinely multidimensional rarefied flows in microchannels[1] , PAUL DELLAR, TIM REIS, Mathematical Institute, University of Oxford — We present lattice Boltzmann simulations of rarefied slip flows driven by applied pressure differences across microchannels of finite length. We correctly capture the nonlinear streamwise pressure variation and the cross-channel velocity component, as well as the streamwise velocity and volume flux. The former effects are both absent from almost all previous work that approximated the pressure difference using a uniform body force. We demonstrate second-order convergence of both velocity components towards the asymptotic solution for long microchannels, and slower convergence of the pressure. We use the standard lattice Boltzmann formulation that reduces to a second-order recurrence relation for the streamwise velocity in uniform shear, and whose analytical solution gives a parabolic profile from wall to wall. We therefore cannot capture Knudsen boundary layers, but instead implement Maxwell–Navier slip boundary conditions directly on the hydrodynamic moments of our discrete velocity model. Our only parameter is the tangential momentum accommodation coefficient, so we require no fitting to known solutions. Our moment-based approach shows that existing boundary conditions impose conditions on higher non-hydrodynamic moments rather than on the tangential fluid velocity itself.

[1]Supported by Award No. KUK-C1-013-04 made by King Abdullah University of Science and Technology (KAUST) and by EPSRC grant EP/E054625/1

Sunday, November 18, 2012 8:00AM - 9:44AM –

Session A28 Free-Surface Flows I 32A - Chair: Eduardo Ramos, Center for Energy Research, National Autonomous University of Mexico

8:00AM A28.00001 The Rotating Polygon Instability of a Swirling Free Surface Flow , TOMAS BOHR, LAUST TOPHØJ, Physics Department and Center for Fluid Dynamics, Technical University of Denmark, JEROME MOUGEL, DAVID FABRE, Institut de Mecanique des Fluides de Toulouse — We present a theory of the rotating polygon instability on a swirling fluid surface [G. H. Vatistas, J. Fluid Mech. **217**, 241 (1990), Jansson et al., Phys. Rev. Lett. **96**, 174502 (2006)]. Our approach is based on potential flow theory, linearised around a potential vortex flow with a free surface. Limiting our attention to the lowest order wave-modes, we obtain an analytically solvable model showing the symmetry breaking instability, and which, together with estimates of the circulation based on angular momentum balance, reproduces the main features of the experimental phase diagram. The generality of our arguments implies that the instability should not be limited to flows with rotating bottom (implying singular behaviour near the corners) and indeed we show that we can obtain the polygons transiently in a much simpler way.

8:13AM A28.00002 Interfacial Instabilities in Torsional Flows , CHING-YAO LAI, Dept. of Physics, Nat'l Taiwan University, Y.-T. SUN, Dept. of Physics, Nat'l Taiwan Normal University, C.-C. CHANG, Inst. of Physics, Academia Sinica, Taipei, Taiwan, Y.-Y. CHEN, Dept. of Physics, Nat'l Taiwan University, P. ARRATIA, Dept. of Mechincal Engineering, University of Pennsylvania, J.- C. TSAI, Inst. of Physics, Academia Sinica, Taipei, Taiwan — In this presentation, we report on current findings on morphology of an oil-water interface in a torsional flow produced by rotating the upper lid of a cylindrical tank. Here, the upper half of the tank is filled with silicon oil and the lower half is filled with water. The interface morphology is investigated as a function of the Reynolds number, based on the upper fluid, and of aspect ratio between the cylinder height and radius (AR=H/R). We find that, at moderate AR, raising the rotation rate (or Re) can induce a continuous change of the oil-water interface morphology from a simple hump to a flat top (plateau), and to a double mound before the interface becomes unstable [1]. At high AR, long water threads and fluid break-ups occur either along the central axis or at a non-zero radius, reflecting the location where the upward flow exhibits a maximum. The changes in the morphology of the interface can be linked to the changes in the secondary flow in the upper fluid. The system exhibits a wealth of behaviors, including symmetry breaking, that illustrates not only the relationship between the upward (secondary) flow and the gravitational force, but also the emergence of various flow and interfacial instabilities.

[1] Lai and Ku, 2009 INTEL Int'l Scienand Engineering Fairs

8:26AM A28.00003 Liquid metal stirring by rotating localized magnetic field in a cylindrical container[1] , SERGIO CUEVAS, MICHEL RIVERO, EDUARDO RAMOS, Center for Energy Research, National Autonomous University of Mexico — We study experimentally the flow in a shallow liquid metal layer (GaInSn) driven by an array of small rotating permanent magnets (12.7 cm diameter) located at the bottom of a cylindrical plexiglas container with a diameter of 203.2 cm. The fluid layer has 13 mm and the maximum analyzed rotation frequency is 7 Hz. The explored magnet arrays vary from one single magnet up to five magnets eccentrically located but equidistant at two different fixed radius. The radial velocity component is obtained using Ultrasound Doppler Velocimetry (UDV) and analyzed through the Fast Fourier Transform. The characteristic frequencies of the flow structures are determined and global flow patterns are approximately reproduced. The flow is also analyzed by image processing in those cases where a free surface oscillation appears and these results are compared with those obtained by UDV.

[1]Support from CONACYT through Project 131399 is thankfully acknowledged.

8:39AM A28.00004 Dynamics of interface separating two fluids under AC electric fields , ASGHAR ESMAEELI, Southern Illinois University Carbondale — Direct Numerical Simulations are performed to study dynamics of a horizontal interface separating two fluids under the influence of an AC electric field. A front tracking/finite difference scheme is used, in conjunction with Taylor's leaky dielectric model, to solve the governing electrohydrodynamics equations in both fluids. The interface becomes unstable beyond a critical voltage, going through a transition period, and forming an oscillatory vertical pillar that points from the liquid of higher electric conductivity to the one with a lower conductivity. It is shown that the pillar shape depends on the dielectric properties of the fluids. The correlation between the frequency of the interface oscillation and that of the imposed electric field is found and a parametric study is performed based on the governing nondimensional number of the problems.

8:52AM A28.00005 Numerical Flow Analysis of Planing Boats , KYLE BRUCKER, THOMAS O'SHEA, DOUGLAS DOMMERMUTH, Naval Hydrodynamics Division, Science Applications International Corporation, THOMAS FU, NSWCCD — The focus of this presentation is to describe the recent effort to validate the computer code Numerical Flow Analysis (NFA) for the prediction of hydrodynamic forces and moments associated with deep-V planing craft. This detailed validation effort was composed of two parts. The first part focuses on assessing NFA's ability to predict pressures on the surface of a 10 degree deadrise wedge during impact with an undisturbed free surface. Detailed comparisons to pressure gauges are presented for two different drop heights, 6 inches and 10 inches. Results show NFA accurately predicted pressures during the slamming event. The second part of the validation study focused on assessing how well NFA was able to accurately model the complex multiphase flow associated with high Froude number flows, specifically the formation of the spray sheet. NFA simulations of a planing hull fixed at various angles of roll (0 degrees, 10 degrees, 20 degrees, and 30 degrees) were compared to experiments from Judge (2012). Comparisons to underwater photographs illustrate NFA's ability to model the formation of the spray sheet and the free surface turbulence associated with planing boat hydrodynamics.

9:05AM A28.00006 Nonlinearity, Viscosity and Air-Compressibility Effects on the Helmholtz Resonant Wave Motion Generated by an Oscillating Twin Body in a Free Surface[1] , PALANISWAMY ANANTHAKRISHNAN, Florida Atlantic University — The problem is of practical relevance in determining the motion response of multi-hull and air-cushion vehicles in high seas and in littoral waters. The linear inviscid problem without surface pressure has been well studied in the past. In the present work, the nonlinear wave-body interaction problem is solved using finite-difference methods based on boundary-fitted coordinates. The inviscid nonlinear problem is tackled using the mixed Eulerian-Lagrangian formulation and the solution of the incompressible Navier-Stokes equations governing the viscous problem using a fractional-step method. The pressure variation in the air cushion is modeled using the isentropic gas equation $pV^{\gamma} = \text{Constant}$. Results show that viscosity and free-surface nonlinearity significantly affect the hydrodynamic force and the wave motion at the resonant Helmholtz frequency (at which the primary wave motion is the vertical oscillation of the mean surface in between the bodies). Air compressibility suppresses the Helmholtz oscillation and enhances the wave radiation.

[1]Work supported by the ONR under the grant N00014-98-1-0151

9:18AM A28.00007 Subharmonic surface waves in a horizontally vibrated container , JOSE PEREZ-GRACIA, JEFF PORTER, FERNANDO VARAS, JOSE VEGA, Universidad Politecnica de Madrid — Horizontal vibrations of rectangular containers first produce harmonic waves and then subharmonic waves as the forcing acceleration is increased. A theoretical analysis is performed that provides both the subharmonic instability threshold and the associated patterns. A key ingredient in the theory is an oscillatory bulk flow (OBF) produced by the vibrating container that extends horizontally to a length comparable with the container depth. The OBF involves harmonic temporal oscillations in the vertical pressure gradient at the free surface. It is precisely those pressure gradient oscillations that trigger subharmonic waves, as in the vertical forced Faraday instability. The obtained results compare well with experimental measurements/visualization that will also be presented.

9:31AM A28.00008 The water entry of streamlined bodies[1] , KYLE BODILY, TADD TRUSCOTT, Brigham Young University — We present the results of an experimental study on the effects of nose shape, wetting angle, and impact angle on the water entry of axisymmetric bodies. Forces, velocity and trajectory are inferred from an inertial measurement unit embedded into the tail and validated by high-speed imaging. A hydrophone is used to record the sounds of cavity collapse to extract a unique signature for each nose shape. Horizontal motion is strongest when impacting with small oblique angles normal to the free surface and weakest when the surface of the body is coated in a half-hydrophobic and half-hydrophilic coating. Additionally, the nose shape has the largest effect on altering acoustic signature for impacts normal to the free surface.

[1]Supported by ONR ULI Maria Medeiros # N000141110872.

Sunday, November 18, 2012 8:00AM - 9:44AM –

Session A29 Porous Media Flows I 32B - Chair: Ruben Juanes, Massachusetts Institute of Technology

8:00AM A29.00001 Nonlinear Taylor dispersion in gravity currents in porous media , MICHAEL SZULCZEWSKI, RUBEN JUANES, Massachusetts Institute of Technology — Taylor dispersion describes how a non-uniform flow can accelerate diffusive mixing between fluids by elongating the fluid-fluid interface over which diffusion acts. While Taylor dispersion has been extensively studied in simple systems such as Poiseuille and Couette flows, it is poorly understood in more complex systems such as porous-media flows. Here, we study Taylor dispersion in porous media during a gravity-driven flow using theory and simulations. We consider a simple geometry for physical insight: a horizontal, confined layer of permeable rock in which two fluids of different densities are initially separated by a vertical interface. We show that the flow exhibits a non-uniform velocity field that leads to Taylor dispersion at the aquifer scale. Unlike the classical model of Taylor dispersion, however, the diffusive mixing is coupled to the flow velocity because it reduces the lateral density gradient that drives the flow. This coupling causes the flow to continually decelerate and eventually stop completely. To model the flow, we develop a non-linear diffusion equation for the concentration of the more dense fluid, which admits an analytical similarity solution. We discuss applications of the model to CO_2 sequestration.

8:13AM A29.00002 Flow through a free-moving porous cylinder within a rotating cylindrical vessel[1] , MOHIT P. DALWADI, SARAH L. WATERS, University of Oxford — Tissue engineering aims to repair or replace damaged body tissue via the engineering of artificial tissues. One method is to seed cells onto a porous biomaterial construct which is then cultured within a rotating bioreactor. We investigate a rotating high-aspect ratio vessel bioreactor that contains a free-moving porous tissue construct. We extend the work of Cummings and Waters [2007], who considered a solid tissue construct, by coupling a single-phase flow external to the tissue construct (modeled by the Navier-Stokes equations) to two-phase flow through the porous tissue construct (modeled using Darcy's equations) via appropriate boundary conditions. We study two flow regimes, corresponding to *near-to* and *far-from* rigid body rotation. We determine the fluid flow through the system for a given construct trajectory. By considering a force balance to deduce the construct trajectory, we obtain a full description of the flow behaviour and the fluid particle paths. We ascertain the residence time of fluid within the construct and, in the future, this work will enable us to calculate the role of advection in the spatiotemporal nutrient distribution, an important consideration for the tissue growth problem.

[1]Funded by EPSRC

8:26AM A29.00003 Convective Shutdown in a Porous Medium , JOHN LISTER, DUNCAN HEWITT, ITG, DAMTP, University of Cambridge, JEROME NEUFELD, ITG, DAMTP & Department of Earth Sciences & BP Institute, University of Cambridge — Convective flow in a porous medium, driven by a buoyancy source along one boundary, is found in many geophysical and industrial processes, and has recently been investigated in the context of CO_2 sequestration. If the domain is closed then the convective flux soon starts to decrease due to the slow evolution of the average interior density. We reveal a close link between such a "one-sided" shutdown system and the "two-sided" statistically steady Rayleigh–Bénard cell. We present high-resolution numerical simulations of convective shutdown at high Rayleigh number Ra in a two-dimensional porous medium. A simple analytic box model of the shutdown system is constructed, with time-dependent Rayleigh and Nusselt numbers, which is based on measurements of the convective flux from a Rayleigh–Bénard cell (Hewitt *et al.* Phys. Rev. Lett. 2012) and gives excellent quantitative agreement with numerical results. These ideas are generalised to model fluids with a power-law equation of state. The dynamical structure of high-Ra shutdown flow is dominated by vertical columnar flow in the interior, and the evolving horizontal wavenumber $k[Ra(t)]$ of the columns gives extremely good agreement with similar measurements of $k(Ra)$ from the columnar flow in a Rayleigh–Bénard cell.

8:39AM A29.00004 Interfacial Motion and Convective Shutdown , DUNCAN HEWITT, ITG, DAMTP, University of Cambridge, JEROME NEUFELD, ITG, DAMTP, & Department of Earth Sciences & BP Institute, University of Cambridge, JOHN LISTER, ITG, DAMTP, University of Cambridge — We present theoretical, numerical and experimental models of the shutdown of convection in a sealed porous domain that is initially stably stratified in two fluid layers. The equation of state is such that the solution which forms at the interface is more dense than either layer. The resultant convective flux across the moving interface slowly shuts down due to the increase in the average lower-layer density. We examine a variety of physical systems, comprised of either immiscible or miscible fluids. In the latter case, diffusion above the interface has a surprising and significant effect at late times. In each case, we develop theoretical box models, based on a rigid-lid assumption for the moving interface, which compare very well with numerical simulations. We explore the validity of the rigid-lid approximation using numerical simulations and experimental results from a Hele-Shaw cell. These both show that interfacial deformation can significantly increase the convective flux, particularly for miscible fluids. Our results have application to a range of geophysical systems, and are particularly relevant to the long-term stability of geologically sequestered CO_2 in a saline aquifer.

8:52AM A29.00005 Scaling of convective dissolution in porous media[1] , JUAN J. HIDALGO, LUIS CUETO-FELGUEROSO, Massachusetts Institute of Technology, Cambridge MA, USA, JAIME FE, University of A Coruna, A Coruna, Spain, RUBEN JUANES, Massachusetts Institute of Technology — Convective mixing in porous media results from the density increase in an ambient fluid as a substance (a solute or another fluid) dissolves into it., which leads to a Rayleigh-Bènard-type instability. The canonical model of convective mixing in porous media, which exhibits a dissolution flux that is constant during the time period before the convective fingers reach the bottom of the aquifer, is not described by the Rayleigh number Ra [Hidalgo & Carrera (2009), J. Fluid Mech.; Slim & Ramakrishnan (2010), Phys. Fluids]. That suggests that dissolution fluxes should not depend on Ra. However, this appears to be in contradiction with recent experimental results using an analogue-fluid system characterized by a non-monotonic density-concentration curve, which naturally undergoes convection [Neufeld et al. (2010), Geophys. Res. Lett.; Backhaus, Turitsyn & Ecke (2011), Phys. Rev. Lett.]. Here we study the scaling of dissolution fluxes by means of the variance of concentration and the scalar dissipation rate. The fundamental relations among these three quantities allow us to study the canonical and analogue-fluid systems with high-resolution numerical simulations, and to demonstrate that both the canonical and analogue-fluid systems exhibit a dissolution flux that is constant and independent of Ra. Our findings point to the need for alternative explanations of recent nonlinear scalings of the Nusselt number observed experimentally.

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

9:05AM A29.00006 Coarsening dynamics of 3D convective dissolution in porous media , XIAOJING FU, LUIS CUETO-FELGUEROSO, RUBEN JUANES, Massachusetts Institute of Technology — Dissolution by convective mixing is an essential trapping mechanism during CO_2 sequestration in deep saline aquifers. Dissolution of the buoyant supercritical CO_2 into the underlying brine leads to a local density increase initially. The resulting CO_2-brine mixture is denser than the two initial fluids, leading to a Rayleigh-Bénard type instability, which greatly accelerates the dissolution process. Both bench-scale experiments and high-resolution computer simulations have shown the initiation and nonlinear interaction of gravity fingers during this density-driven convection process. While 2D analyses of this phenomenon have elucidated important aspects of the dominant flow mechanisms, fundamental issues regarding the coarsening of gravity fingers remain unclear from these studies. In this work, we present high-resolution, 3D simulations of convective dissolution. We observe a previously unreported phenomenon of self-organization of fingers that form coherent network structures in the top boundary layer. Based on this network pattern, we study the coarsening dynamics of convective dissolution in 3D.

9:18AM A29.00007 Chaotic Advection in a Bounded 3-Dimensional Potential Flow , GUY METCALFE, CSIRO Materials Science & Engineering, LACHLAN SMITH, DANIEL LESTER, CSIRO Mathematics, Informatics & Statistics — 3-dimensional potential, or Darcy flows, are central to understanding and designing laminar transport in porous media; however, chaotic advection in 3-dimensional, volume-preserving flows is still not well understood.[1] We show results of advecting passive scalars in a transient 3-dimensional potential flow that consists of a steady dipole flow and periodic reorientation. Even for the most symmetric reorientation protocol, neither of the two invarients of the motion are conserved; however, one invarient is closely shadowed by a surface of revolution constructed from particle paths of the steady flow, creating in practice an adiabatic surface. A consequence is that chaotic regions cover 3-dimensional space, though tubular regular regions are still transport barriers. This appears to be a new mechanism generating 3-dimensional chaotic orbits. These results contast with the experimental and theoretical results for chaotic scalar transport in 2-dimensional Darcy flows.[2,3]

[1]Wiggins, J. Fluid Mech. **654** (2010).
[2]Metcalfe et al, Phil. Trans. R. Soc. **A368** (2010a,b).
[3]Lester et al, Phys. Rev. **E80** (2009), **E81** (2010).

9:31AM A29.00008 Galerkin Dynamical Modeling of Porous Medium Convection using an Eigenbasis from Upper Bound Theory , BAOLE WEN, University of New Hampshire, NAVID DIANATI, University of Michigan, GREG CHINI, University of New Hampshire, CHARLES DOERING, University of Michigan — Galerkin projection is a common strategy for generating ODE models of PDE systems, either as a means of performing highly resolved direct numerical simulation (DNS) or as a method for generating reduced-order dynamical models. Popular bases for the associated spectral expansions include Fourier and Chebyshev modes, eigenfunctions of linear stability operators about "laminar" base solutions or empirical mean flows, or modes arising from Proper Orthogonal Decomposition (POD) of numerical or experimental system realizations. Here we employ an alternative, fully a-priori spectral basis composed of eigenfunctions from upper bound theory, for both fully resolved and reduced dynamical modeling of porous medium convection – a system that is receiving increased attention owing to applications in CO2 sequestration in terrestrial aquifers. Because this new basis is naturally adapted to the dynamics at a given Ra, our DNS requires a fraction of the total number of modes used in traditional (e.g. Fourier) spectral Galerkin simulations. Moreover, for "moderate" Rayleigh numbers ($Ra \lesssim 10^3$) we demonstrate that mode slaving can be used to further reduce the dimension of the truncated dynamical systems.

Sunday, November 18, 2012 8:00AM - 9:44AM –

Session A30 Nanofluids: Experiments 33A - Chair: Alessandro Siria, University of Lyon

8:00AM A30.00001 Fluid-Induced Nanomechanical Fluctuations of an Elastic Membrane[1] , CHARLES LISSANDRELLO, VICTOR YAKHOT, KAMIL L. EKINCI, Boston University — We study the mechanical fluctuations of an elastic membrane induced by collisions with surrounding gas molecules. The system under study is a mm^2 200-nm-thick silicon nitride membrane under tension. The membrane has well-separated resonant modes, which are accurately described by elasticity theory. The membrane is held in a vacuum chamber, and the frequency spectrum of its fluctuations are monitored, as a function of gas pressure, using a sensitive heterodyne interferometer. Our measurements, combined with the fluctuation-dissipation theorem, provide insight into solid-fluid interactions at the nanoscale.

[1]Support from the US NSF (through grants ECCS-0643178, CMMI-0970071, and DGE-0741448) is gratefully acknowledged.

8:13AM A30.00002 Focused electrojetting for nanoscale 3-D fabrication , MINHEE LEE, HO-YOUNG KIM, Seoul National University — Although extreme miniaturization of components in integrated circuits and biochemical chips has driven the development of various nanofabrication technologies, three-dimensional fabrication of nanoscale objects is still in its infancy. Here we propose a novel method to fabricate a free-standing nanowall by the line-by-line deposition of electrospun polymer nanofibers. We show that the electrified nanojet, which tends to get unstable as traveling in free space due to the Coulombic repulsion, can be stably focused onto a narrow line of metal electrode. On the conducting line, the polymer nanojet is spontaneously folded successively to form a wall-like structure. We rationalize the period of spontaneous folding by balancing the tension in the polymer fiber with the electrostatic interaction of the fiber with the metal ground. This novel fabrication scheme can be applied for the development of three-dimensional bioscaffolds, nanofilters and nanorobots.

8:26AM A30.00003 In situ SAXS measurement of nanoparticles filtered with a thin film of macromolecules[1] , F.J. DE JONG, Institute of Multiphase Flows (TUHH), A. BUFFET, G. HERZOG, M. SCHWARTZKOPF, J. PERLICH, German Electron Synchrotron (DESY), Hamburg, V. KOERSTGENS, Physik Dep. E13, TU München, M. MECKLENBURG, T. SCHNOOR, Institute of Polymer Composites (TUHH), P. MUELLER-BUSCHBAUM, Physik Dep. E13, TU München, S. ROTH, German Electron Synchrotron (DESY), Hamburg, K. SCHULTE, Institute of Polymer Composites (TUHH), M. SCHLUETER, Institute of Multiphase Flows (TUHH) — Nanofluidics is connected to many different domains in technology, biology and medicine [1]. Exploring new science using controlled regular nanostructures is by far the most significant benefit of nanofluidics [2]. Macromolecules typically serve as well-ordered nanostructures that can be used for filtering purposes. Especially sieving and filtering of nanoparticles is of big interest in medicine [3]. We investigated the filtering of a colloidal suspension by a thin film of functionalized macromolecules in a microchannel using the technique of Small-angle X-ray scattering (SAXS). SAXS is a nonintrusive measurement technique that enables in situ investigations of the interaction between the functionalized macromolecules and the flowing nanoparticles on the nanoscale. We present our findings on the filtered nanoparticles by the thin film of macromolecules and give an outlook on how to optimize the filtering ability of the thin film based on the in situ SAXS measurements.
[1] Eijkel, *et al.* Micro. Nano. 1, 249 (2005). [2] Schooch, *et al.* Rev. mod. Phy., 80, 840 (2008). [3] Chen *et al.,* Small 7, 1061 (2011).

[1]We acknowledge the Max-Buchner Research Foundation for the financial support for project 2936.

8:39AM A30.00004 Carbon Nanotube Micro-Needles for Rapid Transdermal Drug Delivery[1] , BRADLEY LYON, ADRIANUS INDRAT ARIA, California Institute of Technology, AMIR GAT, Technion - Israel Institute of Technology, JULIA COSSE, LAUREN MONTEMAYOR, MASOUD BEIZAIE, MORTEZA GHARIB, California Institute of Technology — By catalyst patterning, bundles of vertically-aligned carbon nanotubes (CNT) can be assembled to create 2D arrays of hollow micro-needles with feature size as small as a few microns. For transdermal drug delivery, the most challenging mechanical requirement is to make the CNT micro-needle small enough so that delivery is painless yet large enough so that the micro-needle can achieve skin penetration. By taking advantage of capillary action and the nanoporosity of CNT bundles, we can wick high strength polymer into the inter-spacing between nanotubes to augment the stiffness of our micro-needles. For low viscous polymers, the large ratio between the micron sized center hole of the micro-needle and the nanopores of the surrounding CNT allow us to wick polymer through the nanotubes while maintaining an open central hole for drug transport. For a transdermal patch prototype with a delivery area less than 1cm x 1cm square, we can fabricate 50 CNT micro-needles that produces a total flow rate up to 100 uL/s with actuation pressure provided by a mere finger tap. From in vitro experiments, we will demonstrate that CNT micro-needles provide a much faster convective delivery of drugs than conventional topical diffusion based patches.

[1]We acknowledge Zcube s.r.l for their support of this work.

8:52AM A30.00005 Brownian Diffusion of Nanoparticles in Confined Geometries[1] , SHAHRAM POUYA, MANOOCHEHR KOOCHESFAHANI, Michigan Sate University — The transport and motion of nanoparticles is an important aspect of designing micro and nano fluidic devices for biological and chemical analysis. We present preliminary measurements of hydrodynamics of nanoparticles in micro- and nano-confined geometries. Brownian fluctuation of nanoparticles is investigated by imaging the motion of single nanoparticles inside nano/micro gaps at different confinement ratios. The results are presented for the case of zero shear rate, where only pure diffusive motion of nanoparticles within the gap is considered. Results are compared with previous measurements using a different approach and models of this process.

[1]This work was supported by the CRC Program of the National Science Foundation, Grant Number CBET-1033662.

9:05AM A30.00006 How surface functional groups influence fracturation in nanofluids droplets dry-outs[1] , DAVID BRUTIN, FLORIAN CARLE, Aix-Marseille University - UMR 7343 IUSTI Laboratory — We report an experimental investigation of the drying of a deposited droplets of nanofluids with different surface functional groups. For identical nano-particles diameter, material and concentration, identical drying conditions, the substrate and the functional groups at the nano- particles surface are changed. Both flow motion, adhesion, gelation and fracturation occur during the evaporation of this complex matter leading to different final typical patterns. The differences in between the patterns are explained based on the surface chemical potential. Crack shapes and wavelengths are globally proportional to the electrical charges carried at the nano- particles surface which is a new parameter to implement in existing predicting models. Presently only the colloid concentration and softness and the deposit thickness are used (Allain and Limat, 1995). C. Allain and L. Limat, Regular Patterns of Cracks Formed by Directional Drying of a Colloidal Suspension, Phys. Rev. Lett., 74, 2981-2984 (1995).

[1]The authors gratefully acknowledge the help and the fruitful discussions raised with J.B. Lang.

9:18AM A30.00007 Electrodeless electro-hydrodynamic printing of nano-suspensions for personalized medicines[1] , EZINWA ELELE, YUEYANG SHEN, RAJYALAKSHMI BOPPANA, AFOLAWEMI AFOLABI, ECEVIT A. BILGILI, BORIS KHUSID, New Jersey Institute of Technology — Drop-on-demand (DOD) dosing is a promising strategy for manufacturing of personalized medicines. However, current DOD methods developed for chemically and thermally stable, low-viscosity inks are of limited use for pharmaceuticals due to fundamentally different functional requirements. To overcome their deficiency, we developed an electro-hydrodynamic (EHD) DOD method (Appl Phys Lett 97, 233501, 2010) that operates on fluids of up to 30 Pas over a wide range of droplet sizes, does not require direct contact of a fluid with electrodes and provides a precise control over the droplet volume. As most drugs are poorly water soluble, the use of nanoparticles dispersed in water is a promising method for enhancing the drug dissolution rate and bioavailability. The work demonstrates the EHD DOD ability to print aqueous suspensions of drug nanoparticles on highly-porous polymer films. We present a scaling analysis that captures the essential physics of drop evolution. These results show that EHD DOD offers a powerful tool for the evolving field of pharmaceutical technologies for tailoring medicines to individual patient's needs by printing a vast array of predefined amounts of therapeutics arranged in a specific pattern on a porous film.

[1]The work was supported by NSF Engineering Research Center on Structured Organic Particulate Systems.

9:31AM A30.00008 Experimental Nanofluidics in an individual Nanotube , ALESSANDRO SIRIA, PHILIPPE PONCHARAL, ANNE LAURE BIANCE, REMY FULCRAND, STEPHEN PURCELL, LYDERIC BOCQUET, Laboratoire de Physique de la Matiere Condensee et Nanostructures — Building new devices that benefit from the strange transport behavior of fluids at nanoscales is an open and worthy challenge that may lead to new scientific and technological paradigms. [1-3] We present here a new class of nanofluidic device, made of individual Boron-Nitride (BN) nanotube inserted in a pierced membrane and connecting two macroscopic reservoirs. We explore fluidic transport inside a single BN nanotube under electric fields, pressure drops, chemical gradients, and combinations of these. We show that in this transmembrane geometry, the pressure-driven streaming current is voltage gated, with an apparent electro-osmotic zeta potential raising up to one volt. Further, we measured the current induced by ion concentration gradients and show its dependency on the surface charge.

[1] L. Bocquet, E. Charlaix, Chem. Soc. Rev., 39, 1073 (2010).
[2] M. Majumder et al., Nature 2005, 438, 44.
[3] J.K. Holt et al. Science 2006, 312, 1034.

Sunday, November 18, 2012 8:00AM - 9:44AM –

Session A31 Focus Session: Interfacial Engineering in Thermal-Fluids I 33B - Chair: Neelesh Patankar, Northwestern University

8:00AM A31.00001 Droplet Interaction with Lubricant Impregnated Surfaces , RAJEEV DHIMAN, Department of Mechanical Engineering, MIT, JONATHAN DAVID SMITH, Department of Mechanical Engineering,MIT, SUSHANT ANAND, Department of Mechanical Engineering, MIT, ROBERT COHEN, Department of Chemical Engineering, MIT, GARETH MCKINLEY, KRIPA VARANASI, Department of Mechanical Engineering, MIT — The interaction of water drops with lubricant impregnated surfaces was studied experimentally under static and dynamic conditions. Such surfaces contain microscopic roughness features into which a liquid lubricant is impregnated and held by virtue of capillary forces. The remarkable feature of such a construction is that droplets, immiscible to the lubricant, experience negligible resistance to movement on the surface, provided the system is designed carefully under static and dynamic considerations. We describe these considerations and present their implications to droplet mobility. We also study impacting droplets and observe shedding characteristics that are unique to lubricant impregnated surfaces.

8:13AM A31.00002 ABSTRACT WITHDRAWN –

8:26AM A31.00003 The Leidenfrost transition for drops on micro- and nano-textures , HYUK-MIN KWON, Massachusetts Institute of Technology, JAMES BIRD, Boston University, KRIPA VARANASI, Massachusetts Institute of Technology — When a liquid drop contacts a sufficiently hot surface, the drop can float on its own vapor in a process kwon as the Leidenfrost effect. Although it has been observed that the Leidenfrost transition temperature varies with the physical properties of the heated surfaces, the precise mechanisms that set this transition are still not fully understood. Here, we examine the mechanisms for drops contacting a heated surface with well defined micro and nano-textures. We rationalize our experimental results with a scaling model, and subsequently use this model to make predictions that we test experimentally.

8:39AM A31.00004 ABSTRACT WITHDRAWN –

8:52AM A31.00005 Hotspot Cooling with Self-Propelled Jumping Condensate , XIAOPENG QU, JONATHAN B. BOREYKO, FANGJIE LIU, CHUAN-HUA CHEN[1], Department of Mechanical Engineering and Materials Science, Duke University, Durham, NC 27708 — Dynamic hotspots are prevalent in electronic systems including microprocessors and power electronics with constantly changing computing tasks or payloads. Here, we report a new adaptive hotspot cooling technique that rapidly responds to moving hotspots in a passive manner independent of external forces. The hotspot cooling is based upon the self-propelled jumping of dropwise condensate, which directly returns the working fluid from a superhydrophobic condenser to an opposing superhydrophilic evaporator. The adaptive thermal management is accomplished by the preferential evaporation of water at the hotspots and the rapid jumping return of the condensate across the very short inter-plate distance. The proof-of-concept for this hotspot cooling technique will be demonstrated by the adaptive response to hotspots at increasing heat fluxes.

[1]Corresponding author

9:05AM A31.00006 Direct visualization and self-similarity of contact line depinning , ADAM PAXSON, KRIPA VARANASI, Massachusetts Institute of Technology — We report a novel technique to observe the microscale three-dimensional geometry of the contact line of a drop of water on a superhydrophobic surface as it recedes and depins from roughness features. We measure the local receding contact angle at the base of the capillary bridges at the contact line and find them to be equivalent to the macroscale receding contact angle observed on a chemically equivalent smooth surface, providing experimental validation of the Gibbs criterion at the microscale. We use this technique on a dual-scale hierarchically textured surface and reveal a self-similar depinning mechanism that explains how the geometry of the roughness at each length scale affects the adhesion of the contact line. This mechanism allows us to propose a model for predicting the adhesion force of a macroscopic drop, and we use a tensiometer to experimentally verify the model's applicability to both synthetic and natural surfaces.

9:18AM A31.00007 Biphilicity and Superbiphilicity for Wettability Control of Multiphase Heat Transfer , DANIEL ATTINGER, Iowa State University, AMY RACHEL BETZ, Kansas State University, T. M. SCHUTZIUS, University of Illinois at Chicago, J. JENKINS, C.-J. KIM, University of California, Los Angeles, C.M. MEGARIDIS, University of Illinois at Chicago — Multiphase energy transport, such as in boiling, suggests contradictory requirements on the wettability of the solid surfaces coming into contact with the working fluid. On the one hand, a hydrophobic wall promotes nucleation. On the other hand, a hydrophilic wall promotes water contact and enhances the critical heat flux. An analogous situation appears in the opposite thermodynamic process, i.e. condensation. These apparently contradictory requirements can be accommodated with biphilic surfaces, which juxtapose hydrophilic and hydrophobic regions. Biphilic surfaces were first manufactured in 1964 by Young and Hummel, who sprayed Teflon drops onto a smooth steel surface: they showed enhanced heat transfer coefficient during boiling of water. Our recent work has revisited the manufacturing of biphilic surfaces using micro- and nanofabrication processes (Betz et al. 2010, Schutzius et al. 2012); for instance, we fabricated the first superbiphilic surfaces, which juxtapose superhydrophobic and superhydrophilic areas. Using these surfaces, we measured significant enhancement during pool boiling of both the heat transfer coefficient and the critical heat flux. This enhanced performance can be explained by the inherent ability of the surfaces to control multiphase flow, decreasing nucleation energies and shaping drops, bubbles and jets, to maximize transport and prevent instabilities.

9:31AM A31.00008 Staying dry under water[1] , PAUL JONES, Department of Mechanical Engineering, Northwestern University, EDUARDO CRUZ-CHU, Institute of Computational Science, ETH Zurich, CONSTANTINE MEGARIDIS, Department of Mechanical and Industrial Engineering, University of Illinois, JENS WALTHER, Institute of Computational Science, ETH Zurich and Department of Mechanical Engineering, Tech. University of Denmark, PETROS KOUMOUTSAKOS, Institute of Computational Science, ETH Zurich, NEELESH PATANKAR, Department of Mechanical Engineering, Northwestern University — Lotus leaves are known for their non-wetting properties due to the presence of surface texture. The superhydrophobic behavior arises because of the prevention of liquid water from entering the pores of the roughness. Present superhydrophobic materials rely on air trapped within the surface pores to avoid liquid permeation. This is typically unsustainable for immersed bodies due to dissolution of the air, especially under elevated pressures. Here, molecular dynamics simulations are used to demonstrate the non-wetting behavior of an immersed ten-nanometer pore. This is accomplished by establishing thermodynamically sustained vapor pockets of the surrounding liquid medium. Over 300,000 atoms were used to construct the nanopore geometry and simulate SPC/E water molecules. Ambient pressure was varied along two isotherms (300 K, and 500 K). This approach for vapor-stabilization could offer valuable guidance for maintaining surfaces dry even in a submerged state without relying on trapped air. The approach may be extended to control general phase behavior of water adjacent to textured surfaces.

[1]ISEN support is gratefully acknowledged

Sunday, November 18, 2012 8:00AM - 9:31AM –

Session A32 Focus Session: Vortex Dynamics in Fluid-Structure Interactions I 33C - Chair: Stefan Llewellyn Smith, University of California, San Diego

8:00AM A32.00001 Dynamics of Multiple Cylinders in an Inviscid Fluid , ANTHONY LEONARD, California Institute of Technology, ANDREW TCHIEU, ETH Zurich — We consider a collection of circular cylinders moving in an unbounded inviscid fluid. The force on one particular cylinder will be determined by the accelerations, velocities, positions, and radii of all cylinders. We approximate this interaction by taking, in the vicinity of a given cylinder, only the first two terms in a Taylor series expansion of the velocity field induced by the remaining cylinders, i.e. the induced velocity at the center of the given cylinder plus the linearized induced strainrate field. We then correct the potential for this field to satisfy the no through-flow constraint on the cylinder in question. Using Bernoulli's equation, we then derive a coupled system of equations for the forces on all the cylinders. As a result, the added masses are computed to order a^2/d^2 and the force coefficients, not dependent on accelerations, are computed to order a^3/d^3, where a is a typical cylinder radius and d is a typical separation distance. An additional correction for close interactions is proposed. Several examples of cylinder dynamics will be presented.

8:13AM A32.00002 Seal whisker-inspired circular cylinders reduce vortex-induced vibrations , HEATHER BEEM, MICHAEL TRIANTAFYLLOU, MIT — Recent work [1] shows that the undulatory, asymmetric geometry of harbor seal whiskers passively reduces vortex-induced vibration (VIV) amplitudes to less than 0.1 times the whisker diameter. This reduction holds in frontal flows, but due to the elliptical cross-section of the whisker, flows that approach from large angles of attack generate significant vibrational response. The present study investigates the possibility of extending the vibration reduction to unidirectional bodies, such that flows from all angles cause reduced VIV. A method for developing a new geometry that incorporates the "whisker" features into bodies with uniform, circular cross-section is presented. This geometry and multiple variations on it are fabricated into rigid models. Forces are measured on the models while they undergo imposed oscillations and are towed down a water tank. Contour plots of $C_{L,v}$ show peak VIV amplitudes to decrease as much as 28% from that of a standard cylinder. This result holds promise for applications where vibration reduction is desired, regardless of the angle of oncoming flow.

[1] Beem, H., et al., "Harbor Seal Vibrissa Morphology Reduces Vortex-Induced Vibrations," American Physical Society 24th Annual DFD Meeting, Nov. 2011.

8:26AM A32.00003 An experimental study of vortex induced vibrations of inclined flexible cylinders , BANAFSHEH SEYED-AGHAZADEH, YAHYA MODARRES-SADEGHI, University of Massachusetts — When a flexible cylinder is placed in flow, vortices are shed downstream of the cylinder. If the vortex shedding frequency is close to one of the natural frequencies of the structure, the cylinder undergoes vortex induced vibrations. If the cylinder is placed with an angle with respect to the incoming flow (an inclined flexible cylinder) the flow will have a component normal to the long axis of the cylinder and a component in the axial direction. For small angles of inclination, the influence of the axial flow component can be neglected and the cylinder can be treated as a cylinder placed normal to the flow. In the current research, we investigate the degree to which this assumption is valid experimentally. A flexible cylinder with different values of tension along the cylinder is placed at various angles of inclination in the test section of a recirculating water tunnel. The cylinder is clamped at both ends and the transverse displacement of the cylinder at the first mode of vibration is measured using a non-contacting displacement sensor. These measurements together with the flow visualization results are used to investigate the influence of the axial component on the cylinder response at various angles of inclination.

8:39AM A32.00004 Interaction of vortices with flexible piezoelectric beams[1] , OLEG GOUSHCHA, HUSEYIN DOGUS AKAYDIN, NIELL ELVIN, YIANNIS ANDREOPOULOS, The City College of The City University of New York — A cantilever piezoelectric beam immersed in a flow is 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 output. Maximizing the power output of such an electromechanical fluidic system is a challenge. In order to understand the behavior of the beam in a fluid flow where vortices of different scales are present, an experimental facility was set up to study the interaction of individual vortices with the beam. In our set up, vortex rings produced by an audio speaker travel at specific distances from the beam or impinge on it, with a frequency varied up to the natural frequency of the beam. Depending on this frequency both constructive and destructive interactions between the vortices and the beam are observed. Vortices traveling over the beam with a frequency multiple of the natural frequency of the beam cause the beam to resonate and larger deflection amplitudes are observed compared to excitation from a single vortex. PIV is used to compute the flow field and circulation of each vortex and estimate the effect of pressure distribution on the beam deflection.

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

8:52AM A32.00005 VIV experiments with a semi-immersed vertical flexible cylinder driven by top motion , RODOLFO GONCALVES, GUILHERME FRANZINI, CELSO PESCE, ANDRE FUJARRA, University of Sao Paulo, PEDRO MENDES, Petrobras — An experimental investigation of the VIV phenomenon with a long and flexible cylinder was carried out at a recirculating water channel facility and intends to better understand regarding risers non-linear dynamics. The cylinder total length is L=2520mm with an immersed length L_i=720mm and the diameter D=22.2mm, current velocities up to 0.4m/s are carried out in the immersed portion to promote vortex-shedding and consequently VIV. The mass parameter, m* is close to 10. Prescribed vertical and monochromatic harmonic motions were imposed at the top aiming at causing geometric stiffness modulation and, therefore, eigen frequencies oscillations, together with current excitation. The motion imposed at the top has amplitude A_t/L=1% and was imposed at three frequencies ratios, with respect to the first natural frequency in still water: f_t:$f_{N,1}$=1:3,1:2,1:1. Cartesian displacement time-series were obtained for 44 tracking-points, marked with reflective targets placed all along the model, by using a Qualisys optical motion capture system. As expected, a much richer spectral content is revealed under imposed motions than in its absence, i.e. only current condition. The presence of subharmonic components in the response spectra is analogous to the response of a classic parametrically excited system. It was also observed a marked increase of the maximum response amplitude in the case f_t:$f_{N,1}$=1:1

9:05AM A32.00006 Collision of a flexible filament with a point vortex[1] , SILAS ALBEN, University of Michigan — A flexible filament is attracted to a point vortex when they move together as a coupled system. The point vortex collides with the filament at a finite time, with the separation distance tending to zero as a square root of time. The filament curvature, velocity, and pressure loading diverge near the collision. We explain the power laws describing these behaviors.

[1]Support from NSF-DMS grant 1022619 is acknowledged.

9:18AM A32.00007 Simulation of the structural response of a cable immersed in a uniform flow: comparison of three different methodologies[1] , BRUNO CARMO, RAFAEL GIORIA, ALESSANDRO LIMA, JULIO MENEGHINI, University of Sao Paulo — The structural response of slender structures immersed in a fluid flow are very difficult to predict and costly to calculate because of the nonlinear nature of the flow-structure interaction. The search for less expensive, but yet sufficiently accurate methodologies to carry out these calculations has been subject of intense research in the last decades, mainly due to interest of offshore engineering companies. Here we present a comparison of three different methodologies that can be used to predict the structural response of a flexible cable immersed in a uniform flow. The first is the so-called strip approach, in which two-dimensional simulations of the flow at different points along the cable axis are coupled through the cable motion. In the second approach, the flow simulation is three-dimensional considering only domains of few diameters in the spanwise direction, distributed along the cable axis and coupled through the cable displacements. The third is the full three-dimensional calculation of the cable response, considering the entire flow field. We compare the results obtained for a few different Reynolds numbers and cable tensions, analyzing both the accuracy of the results and computational cost.

[1]We acknowledge the support from FAPESP through grant 2011/00131-2

Sunday, November 18, 2012 10:00AM - 11:55AM –

Session B1 Awards Presentations, followed by the Fluid Dynamics Prize and Corrsin Award Lectures Ballroom 20ABC - Chair: Kenny Breuer, Brown University

10:00AM B1.00001 Welcome, Presentation of Awards and DFD Fellowships –

10:25AM B1.00002 Fluid Dynamics Prize Lecture: The Micromechanics of Colloidal Dispersions , JOHN F. BRADY, California Institute of Technology — What do corn starch, swimming spermatozoa, DNA and self-assembling nanoparticles have in common? They are all (or can be modeled as) "particles" dispersed in a continuum suspending fluid where hydrodynamic interactions compete with thermal (Brownian) and interparticle forces to set structure and determine properties. These systems are "soft" as compared to molecular systems largely because their number density is much less and their time scales much longer than atomic or molecular systems. In this talk I will describe the common framework for modeling these diverse systems and the essential features that any hydrodynamic modeling must incorporate in order to capture the correct behavior. Actually computing the hydrodynamics in an accurate and efficient manner is the real challenge, and I will illustrate past successes and current efforts with examples drawn from the diffusion and rheology of colloids to the "swimming" of catalytic nanomotors.

11:10AM B1.00003 Stanley Corrsin Award Lecture: Quantum fluid flows: the strange things we see in superfluid helium , DANIEL LATHROP, University of Maryland — .

Sunday, November 18, 2012 1:30PM - 2:05PM –

Session C33 Invited Session: Synchronization of Eukaryotic Flagella Ballroom 20A - Chair: Eric Lauga, University of California, San Diego

1:30PM C33.00001 Synchronization of Eukaryotic Flagella[1] , RAYMOND E. GOLDSTEIN, University of Cambridge — From unicellular organisms as small as a few microns to the largest vertebrates on earth we find groups of beating flagella or cilia that exhibit striking spatio-temporal organization. This may take the form of precise frequency and phase locking as frequently found in the swimming of green algae, or beating with long-wavelength phase modulations known as metachronal waves, seen in ciliates and in our respiratory systems. The remarkable similarity in the underlying molecular structure of flagella across the whole eukaryotic world leads naturally to the hypothesis that a similarly universal mechanism might be responsible for synchronization. Although this mechanism is poorly understood, one appealing hypothesis is that it results from hydrodynamic interactions between flagella. In this talk I will describe a synthesis of recent experimental and theoretical studies of this issue that have provided the strongest evidence to date for the hydrodynamic origin of flagellar synchronization. At the unicellular level this includes studies of the beating of the two flagella of the wild type unicellular alga *Chlamydomonas reinhardtii* in their native state and under conditions of regrowth following autotomy, and of the flagellar dominance mutant *ptx1*, which displays unusual anti-phase synchronization. Analysis of the related multicellular organism *Volvox carteri* shows it to be an ideal model organism for the study of metachronal waves.

[1]Supported by BBSRC, EPSRC, ERC, and The Wellcome Trust

Sunday, November 18, 2012 1:30PM - 2:05PM –

Session C34 Invited Session: The Logarithmic Layer of Wall-bounded Turbulent Flows Ballroom 20BC - Chair: Juan Carlos del Alamo, University of California, San Diego

1:30PM C34.00001 The logarithmic layer of wall-bounded turbulent flows[1]

, JAVIER JIMÉNEZ, Universidad Politecnica Madrid — The relatively high Reynolds numbers now available in numerical simulations of wall-bounded turbulence allow us to study the flow dynamics well above the viscous wall layer. In particular, we can now observe, manipulate, analyze, and follow in time, the hierarchy of eddies linked to the range of wall distances usually associated with the logarithmic velocity profile. At sufficiently high Reynolds numbers, most of the velocity drop in a boundary layer happens in that region, and the practical large-eddy simulation of attached turbulent flows requires that the dynamics of those self-similar "logarithmic" eddies should be understood well enough to be able to synthesize boundary conditions well above the viscous layer. This talk surveys what has been learned in the last decade. We will see that the logarithmic layer is populated by several self-similar families of eddies connecting the viscous wall structures with the largest outer ones. They have internal Reynolds numbers of the order of thousands, are themselves turbulent, and are better characterized by their energy and Reynolds stresses than by their vorticity. They can be followed in time in full simulations, or in isolation in small numerical boxes, and their dynamics is dominated by a quasi-periodic bursting cycle with a well-defined life history that has much in common with those observed in other shear flows. Those bursts carry most of the Reynolds stresses, and do not appear to be particularly associated with the wall or with the outer flow. They continue to be created and maintained even when either of those "end points" is strongly disturbed. As an example, we will show that it is possible to synthesize a reasonably normal logarithmic layer by applying artificial turbulent off-wall boundary conditions; the key ingredient seems to be the correct modeling of the change of the length scales with the distance to the wall.

[1]Funded by CYCIT and the ERC.

Sunday, November 18, 2012 2:15PM - 4:25PM –

Session D1 Geophysical: Atmospheric I 22 - Chair: Jan Kleissl, University of California, San Diego

2:15PM D1.00001 Velocity statistics and spectra over a forested site measured with a tall mast

, ANTONIO SEGALINI, HENRIK ALFREDSSON, Linne FLOW Centre, KTH Mechanics, Stockholm, Sweden, EBBA DELLWIK, DTU Wind Energy, Technical University of Denmark, Roskilde, Denmark, JOHAN ARNQVIST, HANS BERGSTRÖM, Department of Earth Sciences, Meteorology, Uppsala University, Uppsala, Sweden — In the large expansion of wind power it becomes necessary to use also non-ideal sites for the placement of turbines. Such sites may have a complex terrain in terms of surface elevation as well as being forested. The atmospheric boundary layer is assumed to be severely different as compared to the one over flat, low-vegetation areas, which changes the mean velocity distribution as well as the turbulence intensity, thereby negatively affecting both the power production and loads on the turbines. In this study we use data from a 140 m tall mast in a forest in South-Eastern Sweden, where a unique measurement campaign with sonic anemometers has been running since November 2010 for 16 months. The sonic anemometers give the three velocity components with a frequency resolution of about 10 Hz. The site is covered by approximately 20 m high trees and a 40 degree sector, representative of an approximately homogeneous forest flow, is selected for the analysis of the velocity statistics. The screening of the results indicates the presence of a constant stress layer up to 3-5 canopy heights from the ground. An evaluation of the turbulence statistics in this layer and the levels above is presented. In addition, the spectra are evaluated and compared with the commonly used turbulence models.

2:28PM D1.00002 Large eddy simulation analysis of thermally stratified atmospheric boundary layer interacting with large wind farms[1]

, ADRIAN SESCU, CHARLES MENEVEAU, Johns Hopkins University — Based on a series of large eddy simulation (LES) studies of interactions between the atmospheric boundary layer (ABL) in neutral conditions and infinitely large arrays of wind turbines, new models for the effective roughness length have been developed (Calaf et al. 2010, PoF). Here we consider stratified ABL interacting with large wind turbine arrays. However, in the case of non-neutral conditions, achieving statistically stationary conditions is challenging since, for example, the heat flux at the ground causes vertical profiles of mean temperature to vary in time. To achieve statistical stationarity we use an artificial source of heat, in a region located above ABL, using a PI controller. Another controller is used to drive the flow within ABL, causing the mean velocity to achieve a prescribed direction at a specified height. A suite of simulations at various resolutions and levels of thermal stratification are presented, and the profiles of horizontally averaged velocity, temperature and turbulent fluxes, with and without wind turbines, are compared with each other. In stable conditions the turbulent heat flux increases when wind turbines are included, but in unstable conditions the turbulent heat flux decreases with increasing stratification.

[1]This research is supported by NSF-AGS-1045189.

2:41PM D1.00003 Collective phenomena in large-eddy simulations of extended wind farms[1]

, RICHARD STEVENS, CHARLES MENEVEAU, Johns Hopkins University — A major issue with respect to the incorporation of large wind farms in power grids is that their power output strongly fluctuates over time. Understanding these fluctuations, especially its spatio-temporal characteristics, is important for the design of the backup power that must be available. The power fluctuations of the turbines depend on the effect of the wakes, created by a prior row of turbines, on the operation of the turbines, the inter-turbine correlations, and the interaction between the turbines and the atmospheric boundary layer (ABL). We analyze the power fluctuations in large eddy simulations of extended wind-parks in the ABL. We consider various aggregates of wind turbines such as the total average power signal, or sub-averages within the wind farm. In particular, we find that the power variations of the total wind park decreases more than one would expect if one assumes the power output of the turbines to be uncorrelated. The non-trivial correlations are due to the interactions between turbines placed down-stream from each other. Surprisingly, the frequency spectra of the total wind-farm output show a decay that follows approximately a -5/3 power-law scaling regime, qualitatively consistent with observations made in field-scale operational wind parks (Apt, 2007).

[1]RS is supported by a "Fellowship for Young Energy Scientists" (YES!) of the Foundation for Fundamental Research on Matter (FOM), which is supported by the Netherlands Organization for Scientific Research (NWO). CM is supported by NSF-CBET 1133800.

2:54PM D1.00004 Wind farms and scalar fluxes over a farmland, a Large Eddy Simulation study

, MARC CALAF, École Polytechnique Fédérale de Lausanne (EPFL), CHAD HIGGINS, Oregon State University, MARC PARLANGE, École Polytechnique Fédérale de Lausanne (EPFL) — Recent numerical studies have shown that when the horizontal dimension of the wind farms exceeds the height of the atmospheric boundary layer by a factor of ten or more, a fully developed Wind Farm Array Boundary Layer (WFABL) develops, and scalar fluxes beneath the wind turbines are increased by about 10%. Additionally, in situ measurements have shown a variation in scalar concentrations close to the surface below large wind farms. Therefore, it seems clear that large wind farms do actively interact with and change the local boundary layer, and the consequence (on scalar flux and concentrations) of this interaction might be non-negligible. Are increases in irrigation required to offset an increase in evapo-transpiration? A geo-spatial analysis of the placement of wind farms relative to irrigated lands revealed that 26% of all wind farms in the US are located above some irrigated agriculture, but this overlap only represents 1% of the total wind farm footprint, thus irrigated lands below wind turbines are highly fragmented and variable. Therefore, a new set of Large Eddy Simulations (LES) with variable surface boundary conditions designed to replicate the simplified agricultural landscape below a typical large wind farm were performed. Wind turbines were modeled using the standard actuator disk approach. Results showing the breakup of the internal farmland boundary layer due to the presence of a large wind turbine array will be presented. The role that spatial variability plays in the scalar transport below and above the wind turbines is explored.

3:07PM D1.00005 Energetic Large-Scale Structures found in Urban-type Boundary Layers[1] , BRUNO MONNIER, CANDACE WARK, BRIAN SHAW, MARIA CORTES, SRIHARSHA KANDALA, DIETMAR REMPFER, Illinois Institute of Technology — Flow through a complex environment typical of that found in urban boundary layers is investigated. MAVs maneuvering through an urban field are particularly susceptible to the turbulence encountered in these complex flows. We will present results describing the energetic large-scale coherent structures that are of interest to the aerodynamics community for the purpose of designing the next generation of MAVs. Experimentally we have used SPIV to fully characterize the mean 3D structures as well as investigate various flow regimes encountered in typical urban environments. Various coherent structures that are a function of the geometry of the modeled urban area as well as the incidence angle of the incoming wind direction will be presented and compared with the numerical simulations shown by Kandala et al. An attempt to quantify more accurately the size, location and strength of these structures will be presented.

[1]Supported by AFOSR FA9550-11-1-0056 and the Illinois NASA Space Grant Consortium.

3:20PM D1.00006 Insight on Gust/Turbulence Characteristics in an Urban-type Boundary Layer[1] , CANDACE WARK, BRUNO MONNIER, MARIA CORTES, BRIAN SHAW, SRIHARSHA KANDALA, DIETMAR REMPFER, Illinois Institute of Technology — There is lack of accurate data regarding wind gust profiles in complex environments typical of urban boundary layers. SPIV measurements are used to obtain a 3D map of the mean and turbulent flow encountered in a simplified urban array in a modeled atmospheric boundary layer facility. Gust intensity as a function of space in addition to the gust correlations will be shown and compared with the numerical simulations presented in Kandala et al. The dependence of the gust characteristics on the incidence angle of the incoming wind direction into the urban array is also investigated. These results are of importance to MAV applications as it directly impacts their flying characteristics and can be directly related to surging/plunging/pitching motion studies. The gust characteristics are used to build histograms of the angle of attack that an MAV would encounter while maneuvering through an urban array.

[1]Supported by AFOSR FA9550-11-1-0056 and the Illinois NASA Space Grant Consortium.

3:33PM D1.00007 Numerical Simulations of the Urban-Type Boundary Layer Experiment[1] , SRIHARSHA KANDALA, DIETMAR REMPFER, MONNIER BRUNO, CANDACE WARK, Illinois Institute of Technology, Chicago — Due to their small size, limited power and relatively low speeds compared to the magnitude of typical velocity fluctuations in the atmospheric boundary layer, MAVs are highly susceptible to gusts encountered in complex urban environments. As such, an understanding of the spatial and temporal characteristics of these flow fields can play an important role in the design of these MAVs. In this talk, we present the results of numerical simulation of an urban-type boundary layer described in the talk by Monnier, Wark *et al.* Specsolve, a parallel spectral element solver, is used for these simulations. POD is used to generate a low dimensional representation of the velocity field from the simulation data. Gust statistics and various structures relevant to MAV navigation are presented. These results are compared with SPIV data from the experiment.

[1]This work was supported by AFOSR FA9550-11-1-0056.

3:46PM D1.00008 An Improved Simulation of the Diurnally Varying Street Canyon Flow , NEDA YAGHOOBIAN, JAN KLEISSL, University of California San Diego, KYAW THA PAW U, University of California Davis — The impact of diurnal variation of temperature distribution over building and ground surfaces on the wind flow and scalar transport in street canyons is numerically investigated using the PArallelized LES Model (PALM). The Temperature of Urban Facets Indoor-Outdoor Building Energy Simulator (TUF-IOBES) is used for predicting urban surface heat fluxes as boundary conditions for a modified version of PALM. TUF-IOBES dynamically simulates indoor and outdoor building surface temperatures and heat fluxes in an urban area taking into account weather conditions, indoor heat sources, building and urban material properties, composition of the building envelope (e.g. windows, insulation), and HVAC equipment. Temperature (and heat flux) distribution over urban surfaces of the 3-D raster-type geometry of TUF-IOBES makes it possible to provide realistic, high resolution boundary conditions for the numerical simulation of flow and scalar transport in an urban canopy. Compared to some previous analyses using uniformly distributed thermal forcing associated with urban surfaces, the present analysis shows that resolving non-uniform thermal forcings can provide more detailed and realistic patterns of the local air flow and pollutant dispersion in urban canyons.

3:59PM D1.00009 Flushing of a dense pollutant from a square 2D street canyon , NIGEL KAYE, ZAHRA BARATIAN, Clemson University — Experimental results are presented the rate of flushing of a dense pollutant from a square street canyon. Both finite and continuous release source conditions are considered. For Richardson numbers Ri greater than 1 a clear two-layer stratification is observed, whereas for small Ri the canyon remains well mixed. For the finite release experiments, the initial rate of removal of pollutant is well described with an exponential decay function with the non-dimensional decay rate k being a function of Ri. Steady-state measurements from continuous release experiments for $Ri > 1$ showed that the rate of entrainment across the density interface (directly analogous to the exponential decay rate) is almost exactly the same function of Ri, provided the Richardson number is based on the buoyancy of the lower layer and the canyon height. An energy argument can be used to relate $k(Ri)$ to the mixing efficiency of the turbulent shear flow. Our results indicate that there is a peak in the mixing efficiency at $Ri = 1$ above which the mixing efficiency decreases with increasing Ri and a two-layer stratification is observed.

4:12PM D1.00010 Experimental investigation of the flow over periodic hills , CHRISTIAN CIERPKA, SVEN SCHARNOWSKI, MATTHIAS EHLERT, Bundeswehr University Munich, MICHAEL MANHART, Technical University Munich, CHRISTIAN J. KÄHLER, Bundeswehr University Munich — The flow over periodic hills is a common geometry for the validation of numerical flow simulations and an ERCOFTAC test case (Nr. 81). The numerical prediction is quite difficult, since flow separation and reattachment are not fixed in space and time due to the smooth geometry (Fröhlich et al., J Fluid Mech 526, 2005). Furthermore, the separated and fully three-dimensional flow from the previous hill impinges the next hill and results in very complex flow features including turbulent splashing, Taylor-Görtler vortices and a very thin shear layer with developing Kelvin-Helmholtz instabilities (Rapp, Manhart, Exp Fluids 51, 2011). To examine the characteristics of the flow features even close to the wall an experimental PIV investigation with up to single pixel resolution was performed at the existing experimental facility at the TU Munich. To further enhance spatial resolution and measurement precision also advanced particle tracking methods were applied to evaluate the data. The dynamics of the flow features was measured by using time-resolved sampling of the flow. The analysis of the data shows that all relevant scales can be resolved. Therefore a better data set for the validation of numerical simulation is now available.

Sunday, November 18, 2012 2:15PM - 4:12PM –

Session D2 Convection and Buoyancy-Driven Flows II: Rayleigh-Bénard Convection 23A - Chair: Bob Behringer, Duke University

2:15PM D2.00001 Temperature fluctuations in turbulent Rayleigh- Bénard convection for Ra up to 2×10^{14} and Pr $\simeq 0.8$[1]

, XIAOZHOU HE, DENNIS P.M. VAN GILS, EBERHARD BODENSCHATZ, MPI Dynamics and Self-Organization, Goettingen, Germany, GUENTER AHLERS, UCSB, Santa Barbara, USA — We report on measurements of temperature space-time cross-correlation functions $C_T(r, \tau)$ in Rayleigh-Bénard convection (RBC) near the side wall of a cylindrical sample with aspect ratio $\Gamma \equiv D/L = 1.00$ ($D = 1.12$ m was the diameter and $L = 1.12$ m was the height). The results covered the Rayleigh-number range $4 \times 10^{11} \leq Ra \leq 2 \times 10^{14}$ and the Prandtl-number range $0.79 \leq Pr \leq 0.86$. Our results extend previous measurements for a lower Ra range[2] and confirmed the elliptic approximation (EA) of He and Zhang[3] up to $Ra \simeq 10^{14}$. Using the EA, we determined an effective Reynolds number near the transition to the ultimate state of turbulent RBC.[4]

[1]Supported by the Max Planck Society, the Volkswagen Stiftung, the DFD Sonderforschungsbereich SFB963, and NSF grant DMR11-58514.
[2]X. He, G. He, and P. Tong, Phys. Rev. E, **81**, 065303 (2010).
[3]G.-W. He and J.-B. Zhang, Phys. Rev. E, **73**, 055303 (2006).
[4]X. He, D. Funfschilling, H. Nobach, E. Bodenschatz and G. Ahlers, Phys. Rev. Lett., **108**, 024502 (2012).

2:28PM D2.00002 Logarithmic temperature profiles in turbulent Rayleigh-Bénard convection[1]

, GUENTER AHLERS, UCSB, XIAOZHOU HE, MPIDS Goettingen Germany, DENIS FUNFSCHILLING, LSGC CNRS Nancy, France, DENNIS VAN GILS, EBERHARD BODENSCHATZ, MPIDS Goettingen Germany — We report experimental results for the vertical profiles of the mean temperature $\langle T \rangle$ and the rms temperature fluctuation σ for turbulent Rayleigh-Bénard convection in the interior of a cylindrical sample of aspect ratio $\Gamma \equiv D/L = 0.50$ ($D = 112$ cm and $L = 224$ cm are the diameter and height respectively) over the Rayleigh number range $4 \times 10^{12} \leq Ra \leq 10^{15}$ for a Prandtl number $Pr \simeq 0.8$. We found that $\langle T \rangle$ and σ vary linearly with $ln(z/L)$ where z is the distance from the bottom or top plate. Such a dependence had been predicted[2] for the ultimate state ($Ra > 5 \times 10^{14}$), but was unexpected for the classical state ($Ra < 10^{13}$). The results for $\langle T \rangle$ and σ suggest similarities to the logarithmic profiles found for the velocity in shear flows.[3,4]

[1]Supported by the Max Planck Society, the Volkswagen Stiftung, the DFD Sonderforschungsbereich SFB963, and NSF grant DMR11-58514.
[2]S. Grossmann and D. Lohse, Phys. Fluids **23**, 045108 (2011).
[3]I. Marusic *et al.*, Phys. Fluids **22**, 065103 (2010).
[4]M. Hultmark *et al.*, Phys. Rev. Lett **108**, 094501 (2012).

2:41PM D2.00003 Describing Chaotic Dynamics in Rayleigh-Benard Convection Using Persistent Homology Theory[1]

, JEFFREY TITHOF, MICHAEL SCHATZ, Center for Nonlinear Science and School of Physics, Georgia Institute of Technology, KONSTANTIN MISCHAIKOW, MIROSLAV KRAMAR, VIDIT NANDA, Department of Mathematics, Rutgers University, MARK PAUL, MU XU, Department of Mechanical Engineering, Virginia Polytechnic Institute and State University — We present a new technique for describing the dynamics of spatio-temporal chaos in Rayleigh-Benard convection (RBC). Developed as a tool in algebraic topology, persistent homology theory provides a powerful mathematical formalism for describing the time evolution of geometrical objects. This is done by encoding their topological characteristics in a so-called persistence diagram. When applied to shadowgraph images of spiral defect chaos in RBC, different flow structures correspond to unique features in the persistence diagram. Use of these diagrams helps us to understand the dynamical connections between RBC states, complementing the traditional techniques used in pattern recognition.

[1]This work is supported under NSF grant DMS-1125302.

2:54PM D2.00004 Heat Transport Processes in Turbulent Rayleigh-Bénard Convection described with PDF equations: Numerics and Models

, JOHANNES LUELFF, MICHAEL WILCZEK, Institute for Theoretical Physics, University of Muenster, RICHARD STEVENS, University of Twente, RUDOLF FRIEDRICH, Institute for Theoretical Physics, University of Muenster, DETLEF LOHSE, University of Twente — Rayleigh-Bénard convection, i.e. the convection of a fluid enclosed between two plates that is driven by a temperature gradient, is the idealized setup of a phenomenon ubiquitous in nature and technical applications. Of special interest for this system are the statistics of turbulent temperature fluctuations, which we are investigating for a fluid enclosed in a cylindrical vessel. To this end, we derive an exact evolution equation for the probability density function (PDF) of temperature from first principles. Appearing unclosed terms are expressed as conditional averages of velocities and heat diffusion, which are estimated from direct numerical simulations. Our theoretical framework allows to connect the statistical quantities to the dynamics of Rayleigh-Bénard convection, giving deeper insights into the temperature statistics and transport mechanisms in different regions of the fluid volume, i.e. in the boundary layers, the bulk and the sidewall regions. Furthermore, a minimalistic model of the conditional averages that still incorporates the core features is developed by physical reasoning to highlight the overall character of the heat transport processes.

3:07PM D2.00005 Logarithmic temperature profiles in DNS of turbulent convection

, ROBERTO VERZICCO, Università di Roma "Tor Vergata", RICHARD STEVENS, Dept. Mech. Eng. Johns Hopkins University, DETLEF LOHSE, Physics of Fluids, University of Twente, SIGFRIED GROSSMANN, Dept. of Phys. University of Marburg, UNIROMA2 TEAM, UTWENTE TEAM, MARBURG TEAM — We report numerical results for vertical profiles of mean and rms temperature fluctuations for confined turbulent thermal convection in a cylindrical sample of aspect ratio $\Gamma = 0.5$ and 1 (diameter over height ratio) over a Rayleigh number range $2 \times 10^{10} \leq Ra \leq 2 \times 10^{12}$ and for a Prandtl number $Pr = 0.7$. We found that both quantities vary linearly in $\ln(z)$ with z the distance from the horizontal plates. This behaviour had been predicted for the ultimate regime but it was not expected for the classical state ($Ra \leq 10^{13}$). Similar findings have recently been obtained also experimentally[1] and an excellent agreement with the numerical results has been observed for the mean temperature profiles. The rms fluctuations, in contrast, present relevant differences with respect to the experiments and several explanations are possible.

[1]Ahlers G. et al. "Logarithmic temperature profiles in turbulent Rayleigh-Bénard convection" To appear in Phys. Rev. Lett. 2012.

3:20PM D2.00006 Three-dimensional instability of cylindrical Rayleigh-Benard convection[1] , DE-JUN SUN, BO-FU WANG, Department of Modern Mechanics, University of Science and Technology of China, Hefei, Anhui 230027, China, DONG-JUN MA, School of Aerospace Engineering, Georgia Institute of Technology, Atlanta, GA 30332, USA — The instabilities and transitions of flow in a vertical cylindrical cavity with heated bottom, cooled top and insulated side wall are investigated. The fluid is quiescent at small Rayleigh number and becomes axisymmetric or three dimensional flow when the Rayleigh number is increased. We mainly concerned on the transition of the axisymmetric flow to three dimensional flow through a secondary bifurcation. The steady axisymmetric base flow is obtained by direct numerical simulation and Jacobian-Free Newton-Krylov method, and the stability modes are obtained using the global instability analysis technique. The stability boundaries for the axisymmetric flow are derived for Prandtl numbers from 0.02 to 1 for aspect ratio A (=height/radius) equals 1, 0.9, 0.8, 0.7, respectively. Stable axisymmetric flow beyond the second bifurcation was found in certain ranges of Prandtl number for A=1, 0.9 and 0.8, exclusive of the case for A=0.7. There is no new axisymmetric flow after the second bifurcation for A=0.7 case, but there are multiplicity critical modes as Prandtl number changes., where five kinds of steady modes m=1, 2, 8, 9, 10 and three kinds of oscillatory modes m=3, 4, 6 are presented.

[1]This work was supported by the Natural Science Foundation of China under the Grant No. 11072238 and the 111 Project of China.

3:33PM D2.00007 An experimental study of flow reversals in turbulent Rayleigh-Bénard convection in rectangular cells[1] , SHI-DI HUANG, RUI NI, KE-QING XIA, Department of Physics, The Chinese University of Hong Kong — We present an experimental study of reversals of the large-scale circulation (LSC) in turbulent Rayleigh-Bénard convection. The experiment was conducted in two rectangular cells with the heights and lengths being equal and fixed at 12.6 cm while the widths being 3.84 cm and 2.56 cm, corresponding to lateral aspect ratios Γ being 0.3 and 0.2, respectively. It is found that reversals of the LSC occur more frequently in the $\Gamma = 0.2$ cell than they do in the $\Gamma = 0.3$ cell. The increased temperature fluctuations in the bulk indicates that there are more plumes going through the bulk flow due to the shear effects from the sidewall, which results in a less stable LSC thus more frequent flow reversals.

[1]This work was supported by RGC of Hong Kong SAR (No. CUHK404409 and CUHK403811).

3:46PM D2.00008 Natural convection inside a cylindrical container with a free upper surface , GUILLERMO RAMÍREZ-ZÚÑIGA, GUILLERMO N. HERNÁDEZ, GUILLERMO HERNÁNDEZ-CRUZ, JOSÉ NÚÑEZ, EDUARDO RAMOS, Center for Energy Research - UNAM — This work reports experimental observations and numerical calculations of the natural convective flow inside a cylindrical container (height/diameter= 1.25) with a free upper surface. The bottom and top walls are at high and low temperatures respectively. The Rayleigh number range explored is $10^5 < Ra < 5 \times 10^6$ which includes steady-state and time dependent flows. The working fluid considered is water (Pr=6.67). The observations were made with a stereoscopic PIV system that rotates around the container. With this device, the three component velocity field in the whole volume of the container can be recorded and full three dimensional flow patterns can be reconstructed. The numerical calculation was made with a hybrid finite volume-spectral method considering a free stress boundary for the upper surface. Flow patterns and stability properties are described in the context of potential applications to crystal growth technology.

3:59PM D2.00009 Non- Oberbeck- Boussinesq effects in Poiseuille- Rayleigh- Bénard turbulent channel flow , ALFREDO SOLDATI, Center for fluid Mechanics and Hydraulics, University of Udine, FRANCESCO ZONTA, Dept. Physics, University of Torino — The importance of the Oberbeck-Boussinesq (OB) approximation in turbulent Poiseuille-Rayleigh-Bénard (PRB) flow is established via Direct numerical Simulation (DNS) of water flows with viscosity (μ) and thermal expansion coefficient (β) purely varying with temperature (non-Oberbeck-Boussinesq conditions, NOB). In PRB flows, the combination of buoyancy driven/pressure driven effects produce a complex flow structure, which depends on the relative intensity of the flow parameters (i.e. the Grashof number, Gr, and the shear Reynolds number, Re_τ). In liquids, however, temperature variations induce local changes of fluid properties which influence the macroscopic flow field. We present results for different shear Richardson numbers ($Ri_\tau = Gr/Re_\tau^2$) under constant temperature boundary conditions. As the Richardson number is increased, buoyant thermal plumes are generated. Rising and falling thermal plumes induce large scale thermal convection which increases momentum and heat transport efficiency. Analysis of friction factor (C_f) and Nusselt number (Nu) for NOB conditions shows that the effect of $\mu(T)$ is negligible, whereas the effect of $\beta(T)$ is critical.

Sunday, November 18, 2012 2:15PM - 4:25PM –

Session D3 Multiphase Flows: Bubble Dynamics 23B - Chair: Gretar Tryggvason, Notre Dame University

2:15PM D3.00001 Bubble Dynamics in Bubbly Medium , J. MA, C.-T. HSIAO, G.L. CHAHINE, DYNAFLOW, INC — We present here a two-way coupled euler-lagrange model to study the dynamics Of a primary bubble oscillating in a bubbly mixture. The model simulates the Mixture medium by solving the N-S equations with a moving grid method to Track the motion of the primary bubble wall, while it models the surrounding Small bubbles with the R-P-K-H equation. The two-way coupling between them Is realized through the local mixture density due to the volume change and Motion of the dispersed bubbles. The simulations indicate the surrounding Bubbles absorb the energy radiated from the primary bubble thus reducing Both its maximum radius and period. The dynamics of the surrounding bubbles Result in a phase-shifting between density and pressure waves through the Medium. This is not captured by other analytical solutions assuming Homogeneous medium. Simulations considering the gravity successfully capture The interaction of the small bubbles with and their entrainment into the jet created by the primary bubble. All observations are in good agreements with Experiments of spark generated bubbles in bubbly media.

2:28PM D3.00002 Numerical study of wave breaking and bubble generation in turbulent two-phase Couette flow[1] , DOKYUN KIM, ALI MANI, PARVIZ MOIN, CTR, Stanford University — The objective of the present study is to understand the formation of bubbles due to boundary layer/free-surface interactions. Numerical simulations are performed on a turbulent two-phase Couette flow configuration. A level set method coupled to a subgrid breakup model is used to capture the wave breakup and bubble formation. The free surface is tracked by the level-set method, while small subgrid liquid drops and air bubbles produced from resolved ligaments are transferred from the level-set representation to the Lagrangian representation. The Reynolds and Weber numbers considered are 12,760 and 41,600, respectively, based on water properties. In order to investigate the effect of Froude number on the characteristics of wave breaking and bubble formation, the simulations are done for two different Froude numbers - Fr = 3.9 and 6.8. The statistical data including wave amplitude and bubble size distribution will be presented. The effect of grid resolution on the bubble size distribution will be discussed.

[1]Supported by the Office of Naval Research.

2:41PM D3.00003 Bouncing vs penetration of a particle through fluid interface , ALEX KOTSCH, Department of Engineering Science and Mechanics, Virginia Tech, SUNGYON LEE, Department of Mathematics, UCLA, SUNGHWAN JUNG, Department of Engineering Science and Mechanics, Virginia Tech — Capturing small particles by air bubbles is fundamental to understanding numerous industrial processes of multiphase fluid. In the simple limit of a bubble-sphere interaction, we consider a solid sphere impinging onto a free surface inside the bath of fluid. Experimentally, we observe two main regimes of particle-interface interactions that depend on the initial particle kinetic energy: bouncing and penetration, with a clear transition point from one to the other. Specifically, we find two distinct scalings for change in Weber number versus initial Weber number for the two regimes. In this talk, we present the novel experimental findings and theoretical justifications.

2:54PM D3.00004 Turbulence modulation by microbubbles in channel flow , CRISTIAN MARCHIOLI, University of Udine, DAFNE MOLIN, University of Brescia, ALFREDO SOLDATI, University of Udine — In this paper we examine the mutual interactions between small non-deformable bubbles and turbulence in upward/downward vertical channel flow (at shear Reynolds number Re=150). An Eulerian-Lagrangian approach based on pseudo-spectral direct numerical simulation is used: bubbles are momentum-coupled with the fluid and are treated as pointwise spheres subject to gravity, drag, added mass, pressure gradient, Basset and lift forces. Due to local momentum exchange with the fluid and to the differences in bubble distribution, we observe significant increase (resp. decrease) of wall shear and liquid flowrate in upflow (resp. downflow). We propose a novel force scaling, which can help to judge differences in the turbulence features due to bubble presence. Two-phase flow energy spectra show that bubbles determine an enhancement (resp. attenuation) of energy at small (resp. large) flow scales, a feature already observed in homogeneous isotropic turbulence. Bubble-induced flow field modifications, in turn, alter significantly the dynamics of the bubbles and lead to different trends in preferential concentration and wall deposition. In this picture, the lift force plays a crucial role. We analyze all the observed trends emphasizing the impact that the lift force model has on the simulations.

3:07PM D3.00005 The importance of bubble deformability for strong drag reduction in bubbly turbulent Taylor-Couette flow[1] , DANIELA NAREZO GUZMAN, DENNIS P.M. VAN GILS, CHAO SUN, DETLEF LOHSE, Physics of Fluids Group, Faculty of Science and Technology, J.M. Burgers Center for Fluid Dynamics, and IMPACT Institute, University of Twente, NL — Drag reduction (DR) in two-phase turbulent Taylor-Couette (TC) flow is studied for Reynolds number up to Re $= 2\times10^6$ for pure inner cylinder (IC) rotation, thus extending the previously explored range. DR based on the global torque as a function of the global gas volume fraction (α) over the range 0% up to 4% is obtained. We observe two DR regimes: moderate DR up to 7% for Re $= 5.1\times10^5$ and stronger DR for Re $= 1.0\times10^6$ and 2.0×10^6, remarkably finding more than 40% of DR for $\alpha = 4\%$ at Re $= 2.0\times10^6$. Furthermore, TC flow is locally studied in each regime (Re $= 5.1\times10^5$ and 1.0×10^6) at a fixed $\alpha = 3\%$: statistics of the local liquid flow azimuthal velocity and the local gas concentration are obtained. The local bubble Weber number (We) is computed close to the IC showing that the crossover from the moderate to the strong DR regime occurs roughly at the crossover of $We \sim 1$. We find that a larger local gas volume fraction close to the inner wall has a positive effect on the azimuthal velocity decrease, which is responsible for the observed DR. However for strong DR what is more important for the α values explored here is bubble deformability close to the boundary layer.

[1]Supported by the Technology Foundation STW of the Netherlands.

3:20PM D3.00006 Volume Displacement Effects in Bubble-laden Flows , ANDREW CIHONSKI, JUSTIN FINN, SOURABH APTE, Oregon State University, School of Mechanical, Industrial and Manufacturing Engineering — When a few bubbles are entrained in a traveling vortex ring, it has been shown that even at extremely low volume loadings, their presence can significantly affect the structure of the vortex core (Sridhar & Katz 1999). A typical Euler-Lagrange point-particle model with two-way coupling for this dilute system, wherein the bubbles are assumed subgrid and momentum point-sources are used to model their effect on the flow, is shown to be unable to accurately capture the experimental trends of bubble settling location and vortex distortion for a range of bubble parameters and vortex strengths. The bubbles experience a significant amount of drag, lift, added mass, pressure, and gravity forces. However, these forces are in balance of each other, as the bubbles reach a mean settling location away from the vortex core. Accounting for fluid volume displacement due to bubble motion, using a model termed as volumetric coupling, experimental trends on vortex distortion and bubble settling location are well captured. The fluid displacement effects are studied by introducing a notion of a volumetric coupling force, the net force on the fluid due to volumetric coupling, which is found to be dominant even at the low volume loadings investigated here.

3:33PM D3.00007 On the importance of the Mesler entrainment mechanism in turbulent breaking waves[1] , MILAD MORTAZAVI, VINCENT LE CHENADEC, DOKYUN KIM, ALI MANI, Center for Turbulence Research, Stanford University — Micro bubble generation due to liquid-liquid impact is observed experimentally for simple conditions such as droplet impact (Sigler and Mesler, J. Colloid and Interface Sc., 1989). This regime of air entrainment, called the Mesler regime, is active under certain range of parameters in terms of surface curvature and impact velocity. We have analyzed the importance of the Mesler regime in turbulent breaking waves by employing numerical simulation of a statistically stationary turbulent hydraulic jump at Reynolds number of 88000 and inflow Froude number of 2. A hybrid Lagrangian Eulerian volume of fluid method is used to capture the dynamics of the interface with density ratio of 831. Bubble statistics are compared against the experimental data of Murzyn et al. (Int. J. Multiphase Flow, 2005). Interface structure, curvature, and velocity statistics are analyzed. Our results indicate that impact events in the turbulent hydraulic jump are extremely likely to generate Mesler entrainment.

[1]Supported by the Office of Naval Research.

3:46PM D3.00008 Efficient Time-Stepping Scheme for Incompressible Two-Phase Flows with Large Density Ratios[1] , SUCHUAN DONG, Purdue University — We present an efficient time-stepping scheme within the phase field framework for flows of two immiscible incompressible fluids with large density ratios. The scheme has several attractive characteristics: (1) It is suitable for large density ratios, and numerical experiments with density ratios up to 1000 will be presented; (2) It involves only constant (time-independent) coefficient matrices for all flow variables, which can be pre-computed during pre-processing, so it effectively overcomes the performance bottleneck induced by variable coefficient matrices associated with variable density and variable viscosity; (3) It completely de-couples the computations for all flow variables (velocity, pressure, and phase field function). Numerical simulations will be presented for wall-bounded liquid-gas flows involving large density ratios, moving contact lines, and interfacial topology changes.

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

3:59PM D3.00009 Experimental investigation of the motion of bubble clusters and the flow structures with the clusters , MASANOBU DATE, KAZUKI MAEDA, The University of Tokyo, TOSHIYUKI OGASAWARA, Osaka Prefecture University, SHU TAKAGI, YOICHIRO MATSUMOTO, The University of Tokyo — In upward bubbly flows, mono-dispersed 1 mm spherical bubbles which do not coalesce in the presence of small amount of surfactants in a liquid phase migrate toward the walls due to the shear-induced lift force. Those bubbles form the bubble clusters near the walls [Takagi, S. and Matsumoto, Y., *Annu. Rev. Fluid Mech.* (2011)]. In this study flow structures of the bubbly flow with the bubble clusters and the motion of the bubble clusters are investigated using scanning stereoscopic Particle Image Velocimetry (PIV) and Particle Tracking Velocimetry (PTV), respectively. In order to focus on bubble clusters, 1 mm bubbles are injected near the one of the walls and bubble clusters are formed under some conditions of gas flow rate. From the measurement of the bubbly flows by stereoscopic PIV, it is shown that the bubbles near the wall accelerate surrounding liquids due to their buoyancy and reduce Reynolds stress with increasing a void fraction. Three-dimensional velocity fields are also measured by scanning stereoscopic PIV, and the effect of the bubble cluster on the instantaneous flow fields are analyzed. The results are discussed in the presentation.

4:12PM D3.00010 DNS of Bubbly Flows in Vertical Channels[1] , JIACAI LU, Worcester Polytechnic Institute, SADEGH DABIRI, GRETAR TRYGGVASON, University of Notre Dame — The dynamics of bubbles in upflow, in a vertical channel, is examined using direct numerical simulations (DNS), where both the flow and the bubbles are fully resolved. Two cases are simulated. In one case all the bubbles are of the same size and sufficiently small so they remain nearly spherical. In the second case, several of the small bubbles are coalesced into one large bubble. In both cases lift forces drive small bubbles to the wall, forming a bubble rich wall-layer, removing bubbles from the channel interior until the two-phase mixture there is in hydrostatic equilibrium. The same evolution has been seen in earlier DNS of bubbly upflows, but here the friction Reynolds number is higher (Re^{+}=250). The results show clustering of bubbles in the wall-layer and we examine the mechanism responsible for the clustering and identify how bubbles move in and out of the wall-layer. The dynamics of the bubbles in the channel core is compared with results obtained in fully periodic domains and found to be similar. The presence of the large bubble disrupts the wall-layer slightly, but does not change the overall picture much, for the parameters examined here.

[1]Research supported by DOE (CASL).

Sunday, November 18, 2012 2:15PM - 4:25PM –

Session D4 Drops II 23C - Chair: Osman Basaran, Purdue University

2:15PM D4.00001 Collision and coalescence of liquid drops in a dynamically active ambient fluid , KRISHNARAJ SAMBATH, School of Chemical Engineering, Purdue University, HARIPRASAD SUBRAMANI, Chevron Energy Technology Company, OSMAN BASARAN, School of Chemical Engineering, Purdue University — The fluid dynamics of the collision and coalescence of liquid drops has intrigued scientists and engineers for more than a century owing to its ubiquitousness in nature, e.g. raindrop coalescence, and industry, e.g. breaking of emulsions in the oil and gas industry. The complexity of the underlying dynamics, e.g. occurrence of hydrodynamic singularities, has required study of the problem at different scales – macroscopic, mesoscopic and molecular – using stochastic and deterministic methods. In this work, we adopt a multiscale, deterministic method to simulate the approach, collision, and eventual coalescence of two drops where the drops as well as the ambient fluid are incompressible, Newtonian fluids. The free boundary problem governing the dynamics consists of the Navier-Stokes system and associated initial and boundary conditions that have been augmented to account for the effects of disjoining pressure as the separation between the drops becomes of the order of a few hundred nanometers. This free boundary problem is solved by a Galerkin finite element-based algorithm. The approach and results to be reported build on earlier work by Leal and coworkers, and are used to identify conditions conducive for coalescence in terms of flow and fluid properties.

2:28PM D4.00002 PIV-Analysis of collapsing toroidal droplets[1] , EKAPOP PAIRAM, ERIC BERGER, ALBERTO FERNANDEZ-NIEVES, Georgia Institute of Technology, GEORGIA TECH TEAM — Toroidal droplets are unstable and always undergo a transformation into spherical droplets driven by surface tension. They either break ala Rayleigh-Plateau if the torus is thin or grow fatter to become a single spherical droplet if the torus is fat. We analyze the velocity field inside and outside the toroidal droplet as it transforms into spherical droplets using the particle image velocimetry (PIV) method and compare with recent theoretical calculations for this process.

[1]NSF CAREER

2:41PM D4.00003 Droplet collisions in a liquid , GOSSE OLDENZIEL, RENÉ DELFOS, GERRIT ELSINGA, JERRY WESTERWEEL, Delft University of Technology — The collision of two equally sized fluid droplets in a continouous phase consisting of a second liquid is investigated. The outcome of the collision, i.e. coalescence or bouncing depends, for two given fluids, on the relative velocity of the droplets just before impact, and the mis-alignment of the collision Δx, nondimensionialized in a Weber number, $We = \rho_d U_{rel}^2 d_{ave}/\sigma$ and the alignment parameter $B = \Delta x/d_{ave}$. We studied nearly head-on collisons by launching the droplets from two opposing capillaries. High speed dual-axis shadowgraphy is used to reconstruct the 3D-droplet trajectories. We found that if the Weber number exceeds a critical value, the droplets will coalesce; for lower values they bounce off again. For silicon oil droplets with a dynamic viscosity 4.6 times that of the surrounding phase (water) the critical Weber number is found to be $We \approx 22.5$. For all bouncing droplets, the contact time was measured and found to be approximately equal to the theoretical droplet oscillation frequency period. Current work aims at investigating the influence of external continuous phase turbulence on the outcome of the collisions by placing the two-capillary system in between two counterrotating discs ("von Kármán flow").

2:54PM D4.00004 Simulations of Droplet Coalescence in Simple Shear Flow , OREST SHARDT, JOS DERKSEN, SUSHANTA MITRA, University of Alberta — We present highly-resolved simulations of droplet coalescence in a shear flow. In general, droplet coalescence is difficult to simulate due to the wide range of relevant length scales: the diameter of a droplet may be 10^4 times larger than the minimum thickness of the film between a pair of droplets. In a shear flow, droplets coalesce unless a critical capillary number is exceeded. We found that this critical capillary number is about 20 times higher in simulations than in experiments when the domain size is at the scale of previous work. We use the binary-liquid free-energy lattice Boltzmann method. In this diffuse-interface model, the interface is characterized by a tension, thickness, and diffusivity. Using the parallel computational power of graphics processing units, we simulate sufficiently large domains to determine the dependence of the critical capillary number (Ca) on the diameter (D) of the droplets (relative to the interface thickness). The exponent in this scaling law depends strongly on the interface diffusivity. A new region in the Ca vs D map for the outcome of a simulated droplet collision was identified. We also study geometric effects. Due to the high resolutions that we simulate, we obtain critical capillary numbers that approach experimental values.

3:07PM D4.00005 Wettability effects on droplet coalescence[1] , PERCIVAL GRAHAM, DENNIS DE PAUW, ALI DOLATABADI, Concordia University — Droplet impingement has been studied since 1895, with the works of A.M. Worthington. Throughout the past century, a variety of interesting phenomena have been uncovered. These include the bouncing of droplets off of each other or liquid pools, intricate droplet splashing mechanics, and droplets bouncing off of superhydrophobic surfaces; to name a few. In addition to intricate phenomena, droplet dynamics are relevant to many engineering applications, such as painting, spray coating ink-jet printing, and ice accumulation. These fields all involve interactions between droplets; therefore, studying droplet coalescence would benefit them greatly. The works presented include the coalescence of droplets with different impact conditions, various offsets, and at different wettabilities. Surface wettabilities studied are hydrophilic, hydrophobic and superhydrophobic. Fascinating phenomena observed include, bouncing of the impinging droplet off of the sessile droplet, sliding of the impinging droplet along the sessile droplet, and induced detachment on the sessile droplet on superhydrophobic surfaces. In order to capture the maximum spreading of the merged droplets, models related to coalescence of droplets in air and maximum spreading of a single droplet are combined to yield a new model to predict the maximum spreading of head-on droplet impact. Based on the free surface, and accuracy of the analytical model, droplet impact could be viewed as a mix of droplet coalescence in a gaseous media and droplet impact on a dry surface.

[1]Funding from NSERC

3:20PM D4.00006 Self-Propelled Jumping Drops on Leidenfrost Surfaces: Experiments and Simulations , FANGJIE LIU, Department of Mechanical Engineering and Materials Science, Duke University, Durham, NC 27708, GIOVANNI GHIGLIOTTI, JAMES J. FENG[1], Department of Mathematics, University of British Columbia, Vancouver, BC, Canada V6T 1Z2, CHUAN-HUA CHEN, Department of Mechanical Engineering and Materials Science, Duke University, Durham, NC 27708 — Coalescing drops spontaneously jump on a variety of biological and synthetic superhydrophobic surfaces, with potential applications in self-cleaning materials and self-sustained condensers. To investigate the mechanism of the self-propelled jumping drops, a Leidenfrost collider was constructed on which drops floating on a vapor layer were guided to merge and subsequently jump on a heated substrate. Above a threshold drop diameter, we experimentally observed a constant energy conversion efficiency, which is the ratio of the kinetic energy of the merged drop to the surface energy released upon coalescence. This trend matched with prior reports of jumping condensate drops on a superhydrophobic surface. The capillary-inertial scaling of the jumping process was confirmed with a phase-field simulation of two equally-sized spherical drops coalescing on a flat surface with a contact angle of 180°. The numerical simulation revealed the role of viscous dissipation, leading to reduced energy conversion efficiency when the Ohnesorge number based on the drop diameter approaches unity.

[1]Department of Chemical and Biological Engineering, University of British Columbia, Vancouver, BC, Canada V6T 1Z3

3:33PM D4.00007 Symmetric and Asymmetric Coalescence of Drops on a Substrate , FEDERICO HERNANDEZ-SANCHEZ, LUUK LUBBERS, ANTONIN EDDI, JACCO SNOEIJER, Univ of Twente — The coalescence of viscous drops on a substrate is studied experimentally and theoretically. As a result, a universal shape of the bridge between the drops was found. The dynamics of the bridge is accurately described by similarity solutions of the one-dimensional lubrication equation. Such solutions were found for equal and unequal contact angles of the two drops, and characterized by the ratio of contact angles. Our theory predicts a bridge that grows linearly in time and stresses the strong dependence on the contact angles. Without any adjustable parameters, we find quantitative agreement with all experimental observations. The results reveal the importance of asymmetry on the coalescence process.

3:46PM D4.00008 Spontaneous penetration of a non-wetting drop into an exposed pore[1] , PENGTAO YUE, YURIKO RENARDY, Virginia Tech — We consider the penetration process of a liquid drop approaching an exposed pore along the axis of symmetry, which is intended to model the penetration of non-wetting drops into a porous medium. Inertia and gravity are neglected at the current stage. Different from the penetration into a capillary tube in literature, the drop may spread on the outer surface of the porous medium. Based the on the mechanical equilibrium states, we find the critical drop radius, below which the drop penetration is spontaneous. We further identify five penetration regimes based on the drop radius and the static contact angle, all of which are exemplified by phase-field simulations. The free energy as a function of penetration depth reveals only two stable equilibrium states — the drop either enters the pore completely (maximum penetration) or stays at the pore inlet (zero penetration). For a non-penetrating drop radius, the free energy has a local maximum which constitutes an energy barrier that prevents spontaneous penetration. Finally, we modify the Lucas-Washburn equation to describe the dynamic process of penetration, which turns out to greatly overestimate the penetration rate due to the neglect of dissipation from moving contact lines and flows outside the pore.

[1]This work is supported by NSF-DMS 0907788.

3:59PM D4.00009 Drop Impingement on Highly Wetting Porous Thin Films: Theoretical Justification for the Washburn-Reynolds Number , CULLEN BUIE, YOUNG SOO JOUNG, Massachusetts Institute of Technology — Recently we've introduced a dimensionless parameter named the Washburn-Reynolds number (Re_w). The Washburn-Reynolds number predicts drop impingement modes on highly wetting porous thin films. Physically, the Washburn-Reynolds number (Re_w) can be interpreted as the ratio between the inertia of the impinging droplet and capillary transport in the porous thin film. In this talk we outline the theoretical considerations that lead to the Washburn-Reynolds number. To estimate droplet spreading after impact, we've devised an energy conservation expression employing a capillary potential energy term. This term leads to another dimensionless parameter denoted the capillary Weber number (We_c), which is the ratio of the kinetic energy of the droplet to the capillary energy of the surface. The energy equation can be simplified as a function of the Weber number (We) and the dimensionless spreading speed constant (C^*_{cap}) in high and low We_c limits, respectively. Re_w is obtained from the log scale geometric average of We and C^*_{cap}. Our theoretical analysis and experimental results verify that Re_w is useful to predict impingement modes on highly wetting porous films for a wide range of impact velocities and fluid properties.

4:12PM D4.00010 Fluid dynamics following flow shut-off in bottle filling , SUMEET THETE, School of Chemical Engineering, Purdue University, SANTOSH APPATHURAI, HAIJING GAO, Chevron Corporation, OSMAN BASARAN, School of Chemical Engineering, Purdue University — Bottle filling is ubiquitous in industry. Examples include filling of bottles with shampoos and cleaners, engine oil and pharmaceuticals. In these examples, fluid flows out of a nozzle to fill bottles in an assembly line. Once the required volume of fluid has flowed out of the nozzle, the flow is shut off. However, an evolving fluid thread or string may remain suspended from the nozzle following flow shut-off and persist. This stringing phenomenon can be detrimental to a bottle filling operation because it can adversely affect line speed and filling accuracy by causing uncertainty in fill volume, product loss and undesirable marring of the bottles' exterior surfaces. The dynamics of stringing are studied numerically primarily by using the 1D, slender-jet approximation of the flow equations. A novel feature entails development and use of a new boundary condition downstream of the nozzle exit to expedite the computations. While the emphasis is on stringing of Newtonian fluids and use of 1D approximations, results will also be presented for situations where (a) the fluids are non-Newtonian and (b) the full set of equations are solved without invoking the 1D approximation. Phase diagrams will be presented that identify conditions for which stringing can be problematic.

Sunday, November 18, 2012 2:15PM - 4:25PM –

Session D5 Computational Fluid Dynamics II 24A - Chair: Elias Balaras, George Washington University

2:15PM D5.00001 Tweed Relaxation: a new multigrid smoother for stretched structured grids

, THOMAS BEWLEY, Flow Control Lab, Dept of MAE, UC San Diego, ALIREZA MASHAYEKHI, Dept. of Physics, U of Toronto — In DNS/LES of the NSE using a fractional step method, one must accurately solve a Poisson equation for the pressure update at each timestep. This step often represents a significant fraction of the overall computational burden and, when Fourier methods are unavailable, geometric multigrid methods are a preferred choice. When working on an unstretched Cartesian grid, the red-black Gauss-Seidel method is the most efficient multigrid smoother available. When working on a Cartesian grid that is stretched in 1 coordinate direction to provide grid clustering near a wall, zebra relaxation, on sets of lines perpendicular to the wall, is most efficient. When working on a structured grid that is stretched in 2 or 3 coordinate directions, however, one is forced to alternate the directions that the zebra relaxation is applied in order to pass information quickly across all regions of grid clustering. A new relaxation method is introduced which is shown to significantly outperform such alternating direction line smoothers. This new method is implicit along sets of lines that branch and form 90° corners, like the stripes at the shoulder of a tweed shirt, to stay everywhere perpendicular to the nearest wall, thus passing information quickly across all regions of grid clustering.

2:28PM D5.00002 A new expicit projection method for incompressible flows

, SANGRO PARK, CHANG-HOON LEE, Yonsei University — When solving unsteady incompressible flows, the divergence-free condition should be satisfied. For this, the non-linear terms in the Navier-Stokes equation should be projected onto divergence-free space by an operator which arises from taking divergence of the Navier-Stokes equation. The calculation of projecting non-linear fields requires a lot of computational cost because the projection typically relies on an iterative solution of pressure. In this study, we propose an explicit projection method based on the spectral solution of the Poisson equation in the infinite domain and local truncation in the physical space, which does not require iterations. For validations of our methods, we applied the proposed method to the 2-dimensional Taylor-Green vortex simulation and forced isotropic turbulence simulation. The test results show that our method saved computational cost enormously while maintaining reasonable accuracy of flow field. More details about the suggested method and the performance of the method will be discussed in the meeting.

2:41PM D5.00003 Tetrahedralization of Isosurfaces with Guaranteed-Quality by Edge Rearrangement (TIGER)[1]

, SHAWN WALKER, Louisiana State University — We present a method for generating tetrahedral meshes of solids whose boundary is a smooth surface. The method uses a background grid (body-centered-cubic (BCC) lattice) from which to build the final conforming 3-D mesh. The algorithm is fast, robust, and provides useful guaranteed dihedral angle bounds for the output tetrahedra. The dihedral angles are bounded between 8.5 and 164.2 degrees. If the lattice spacing is smaller than the "local feature size," then the dihedral angles are between 11.4 and 157.6 degrees (c.f. Labelle, Shewchuk 2007). The method is simple to implement and performs *no* extra refinement of the background grid. The most complicated mesh transformations are 4-4 edge flips. Moreover, the only parameter in the method is the BCC lattice spacing. Applications of the method range from free boundary flows, to modeling deformations, shape optimization, and to anything that requires dynamic meshing such as virtual surgery. A MATLAB demonstration will be given to show case the method.

[1]Research Funded by NSF grant DMS-1115636

2:54PM D5.00004 On a robust ALE method with the discrete primary and secondary conservation properties[1]

, SEONGWON KANG, NAHMKEON HUR, Dept. of Mechanical Engineering, Sogang University, Korea — The objective of the present study is to construct a robust, implicit discretization method for the arbitrary Lagrangian-Eulerian (ALE) method for deforming grids. In order to minimize the effect of an artificial diffusion, we present a novel implicit method derived using the secondary conservation property enforced in both spatial and temporal discretization. When applied to the Navier-Stokes equation, the proposed method satisfies conservation of the discrete mass, momentum and kinetic energy in both incompressible and compressible flows. We compared the different choices for discretization in the ALE method by an analysis of the truncation errors. With the numerical tests using the cases with high Reynolds numbers, an improved stability was observed using the revised discretization method compared to the existing methods.

[1]Supported by the Korea government (MEST) via M3CFD ERC center

3:07PM D5.00005 Wavelet-based adaptive numerical simulation of unsteady 3D flow around a bluff body

, GIULIANO DE STEFANO, University of Naples (ITALY), OLEG VASILYEV, University of Colorado at Boulder — The unsteady three-dimensional flow past a two-dimensional bluff body is numerically simulated using a wavelet-based method. The body is modeled by exploiting the Brinkman volume-penalization method, which results in modifying the governing equations with the addition of an appropriate forcing term inside the spatial region occupied by the obstacle. The volume-penalized incompressible Navier-Stokes equations are numerically solved by means of the adaptive wavelet collocation method, where the non-uniform spatial grid is dynamically adapted to the flow evolution. The combined approach is successfully applied to the simulation of vortex shedding flow behind a stationary prism with square cross-section. The computation is conducted at transitional Reynolds numbers, where fundamental unstable three-dimensional vortical structures exist, by well-predicting the unsteady forces arising from fluid-structure interaction.

3:20PM D5.00006 Adaptive Wavelet Collocation Method in Shallow Water Model: Validation Study[1]

, SHANON RECKINGER, Fairfield University, OLEG VASILYEV, BAYLOR FOX-KEMPER, University of Colorado at Boulder — The adaptive wavelet collocation and Brinkman penalization methods are applied to the shallow water model and validated. The wavelet method solves the equations on temporally and spatially varying meshes, which allows a higher effective resolution to be obtained with less computational cost. The grid adaptation is achieved by using the ability of wavelet multiresolution analysis to identify and isolate localized dynamically dominant flow structures, e.g., vortices, and to track these structures on adaptive computational meshes. In addition to studying how the shallow water model behaves on non-uniform, time varying grids, this work also sets out to improve the representation of continental topology through an extension of the Brinkman penalization method. This numerical technique works by altering the governing equations in such a way that no slip boundary conditions are enforced. When coupled with the adaptive wavelet collocation method, the flow near a complex boundary can be well defined. In previous work, the methods were presented, fully derived, and convergence was demonstrated. In this work, a variety of benchmark studies will be presented to validate the model and insight will be given on possible directions for wavelets in ocean modeling.

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

3:33PM D5.00007 On the POD based reduced order modeling of high Reynolds flows

, FARIDUDDIN BEHZAD, BRIAN HELENBROOK, GOODARZ AHMADI, Department of Mechanical and Aeronautical Engineering, Clarkson University, Potsdam, NY, USA — Reduced-order modeling (ROM) of a high Reynolds fluid flow using the proper orthogonal decomposition (POD) was studied. Particular attention was given to incompressible, unsteady flow over a two-dimensional NACA0015 airfoil. The Reynolds number is 10^5 and the angle of attacked of the airfoil is 12°. For DNS solution, hp-finite element method is employed to drive flow samples from which the POD modes are extracted. Particular attention is paid on two issues. First, the stability of POD-ROM resimulation of the turbulent flow is studied. High Reynolds flow contains a lot of fluctuating modes. So, to reach a certain amount of error, more POD modes are needed and the effect of truncation of POD modes is more important. Second, the role of convergence rate on the results of POD. Due to complexity of the flow, convergence of the governing equations is more difficult and the influences of weak convergence appear in the results of POD-ROM. For each issue, the capability of the POD-ROM is assessed in terms of predictions quality of times upon which the POD model was derived. The results are compared with DNS solution and the accuracy and efficiency of different cases are evaluated.

3:46PM D5.00008 Projection of Discontinuous Galerkin Variable Distributions During Adaptive Mesh Refinement , CARLOS BALLESTEROS, MARCUS HERRMANN, Arizona State University — Adaptive mesh refinement (AMR) methods decrease the computational expense of CFD simulations by increasing the density of solution cells only in areas of the computational domain that are of interest in that particular simulation. In particular, unstructured Cartesian AMR has several advantages over other AMR approaches, as it does not require the creation of numerous guard-cell blocks, neighboring cell lookups become straightforward, and the hexahedral nature of the mesh cells greatly simplifies the refinement and coarsening operations. The *h*-refinement from this AMR approach can be leveraged by making use of highly-accurate, but computationally costly methods, such as the Discontinuous Galerkin (DG) numerical method. DG methods are capable of high orders of accuracy while retaining stencil locality—a property critical to AMR using unstructured meshes. However, the use of DG methods with AMR requires the use of special flux and projection operators during refinement and coarsening operations in order to retain the high order of accuracy. The flux and projection operators needed for refinement and coarsening of unstructured Cartesian adaptive meshes using Legendre polynomial test functions will be discussed, and their performance will be shown using standard test cases.

3:59PM D5.00009 A Spectral Adaptive Mesh Refinement Method for Homogenous Isotropic Turbulence , LEILA NASR AZADANI, ANNE STAPLES, Virginia Tech — We present an algorithm for accelerating simulations of homogenous isotropic turbulence. The method is akin to an adaptive mesh refinement (AMR) technique, applied in Fourier space. In direct numerical simulations of turbulence (DNS) the mesh size or the number of Fourier modes is defined based on the ratio of the sizes of the largest to smallest eddies that can be formed during the computation. The range of spatial scales in evolving turbulent flows changes with time. Early in a computation there may exists only large eddies and a coarse mesh will be enough to capture all the details of the flow, while at another time smaller eddies may form and a finer mesh will be required to resolve all scales. Therefore, instead of performing DNS with a constant fine mesh, AMR techniques can be applied and the mesh size can be varied during the computation in order to have optimum mesh sizes and save computational time. The spectral AMR method we present here is applied to 2D and 3D homogenous isotropic turbulence and results are compared with the DNS performed using a fine mesh.

4:12PM D5.00010 ABSTRACT WITHDRAWN –

Sunday, November 18, 2012 2:15PM - 4:25PM –

Session D6 Electrokinetics II 24B - Chair: David Salac, SUNY Buffalo

2:15PM D6.00001 Multiscale simulation of electroosmotic flows , LIN GUO, MARK ROBBINS, Johns Hopkins University, SHIYI CHEN, Johns Hopkins University and Peking University, JIN LIU, Washington State University — We develop an efficient hybrid multiscale method for simulating nano-scale electroosmotic flow based on spatial "domain decomposition" [1]. Molecular dynamics (MD) is used in the near wall region where atomistic details are important. AmultigridParticle-Particle Particle-Mesh (PPPM) method [2] is used to calculate the long-range Coulombic interaction between charged ions. Continuum (incompressible Navier-Stokes) equations for the solvent are solved in the bulk region, reducing the computational cost substantially. A discrete description of ions is retained in the continuum region because of the low density of ions and the long-range of electrostatic interactions. Langevin dynamics is used to model the Brownian motion of these ions in the implicit solvent. The fully atomistic and continuum descriptions are coupled through "constrained dynamics" [1] in an overlap region. Flux of charged and solvent particles between continuum and MD regions is included. Simulation results for different channel sizes are provided. To benchmark this multiscale scheme, we compare results with pure MD simulations.

[1] X. B. Nie, S. Y. Chen, W. N. E, and M. O. Robbins, J. Fluid Mech., 500:55–64, 2004.
[2] J. Liu, M. R. Wang, S. Y. Chen, and M. O. Robbins, J. Comput. Phys., 229:7834-7847, 2010.

2:28PM D6.00002 Colloidal Trapping within a Second Kind Electroosmotic Vortex Pair at a Microchamber-Nanoslot Interface , YOAV GREEN, GILAD YOSSIFON, Technion - Israel Institute of Technology — It has been shown in previous works that due to strong tangential electric fields, electroosmotic vortices of the second kind appear at the interface of a microchamber-narrow nanoslot. Continuity of fluxes causes strong field focusing effects which change the characteristic behavior of the current-voltage curve and traps particles, larger than the nanoslot height, at the nanoslot interface. We solve the steady state Poisson-Nernst-Planck-Stokes equations for a 2D model of a microchamber-narrow nanoslot interface. Afterwards, the particle's dynamic equations of motion including a non-divergence free dielectrophoresis (DEP) force are solved to obtain the particle trajectories. It is demonstrated that due to the short range DEP force particles in the vicinity of the interface are quickly trapped which stands in qualitative agreement with the experimental findings.

2:41PM D6.00003 Effect of Electro-Osmotic Flow on Energy Conversion on Superhydrophobic Surfaces , SESHADRI GOWRISHANKAR, Indian Institute of Technology, Bombay, TOBIAS BAIER, Centre for Smart Interfaces, TU Darmstadt, Germany — It has been suggested that a superhydrophobic surface, by virtue of the presence of no-shear zones, can greatly enhance the transport of surface charges. This would lead to a considerable increase in the streaming potential, a feature which could find possible applications in micro-energy harvesting devices. Such devices are of promise in micro-fluidic studies in view of their ability to act as effective energy conversion devices on the micro-scale. In our paper, we use a theoretical approach to show that the generation of a streaming potential in such superhydrophobic geometris is significantly limited from that otherwise expected because of the current generated from a reverse electro-osmotic flow. We also show that, for large values of free surface charge densities, the electro-osmotic flow current would engender a saturation in both the power extracted and efficiency of energy conversion that is achievable in such systems. Our analysis therefore indicates that fluids with very low conductivity should be preferred in energy conversion devices. Finally, we extrapolate our results to show that a saturation to both the energy conversion and the efficiency would be obtained in all flow geometries, although the charge density of the free surface at which this happens can vary.

2:54PM D6.00004 Electroosmotic Flow of Power-Law Fluids in a Cylindrical Microcapillary , M.H. SAIDI, School of Mechanical Engineering, Sharif University of Technology, ASHKAN BABAIE, Department of Mechanical Engineering, University of British Columbia, ARMAN SADEGHI, School of Mechanical Engineering, Sharif University of Technology, CENTER OF EXCELLENCE IN ENERGY CONVERSION TEAM — In biological applications where most fluids are considered to be non-Newtonian, Newtonian law of viscosity looks insufficient for describing the flow characteristics. In the present work, the electroosmotic flow of power-law fluids in a circular micro tube is investigated. The Poisson-Boltzmann equation for electrical potential is solved numerically in the complete form without using the Debye-Hückel approximation. The physical model includes the Joule heating and viscous dissipation effects. Once the momentum and energy equations are solved numerically, a parametric study is done to investigate the effects of different parameters such as flow behavior index, wall zeta potential and the Debye-Hückel parameter on thermal and hydrodynamic characteristics of the flow. Results show that based on the value of viscous dissipation and the Debye-Hückel parameter the non-Newtonian characteristics of the flow can lead to significant changes regarding to Newtonian behaviors. The provided results in this study would lead to accurate prediction of temperature of biofluids in Lab-on-a-chip devices which is vital for retaining samples in a healthy condition.

3:07PM D6.00005 Measurements of Induced-Charge Electroosmotic Flow Around a Metallic Rod , ALI BESKOK, CETIN CANPOLAT, Old Dominion University — A cylindrical gold-coated stainless steel rod was positioned at the center of a straight microchannel connecting two fluid reservoirs on either end. The microchannel was filled with 1 mM KCl containing 0.5 micron diameter carboxylate-modified spherical particles. Induced-charge electro-osmotic (ICEO) flow occurred around the metallic rod under a sinusoidal AC electric field applied using two platinum electrodes. The ICEO flows around the metallic rod were measured using micro particle image velocimetry (micro-PIV) technique as functions of the AC electric field strength and frequency. The present study provides experimental data about ICEO flow in the weakly nonlinear limit of thin double layers, in which, the charging dynamics of the double layer cannot be presented analytically. Flow around the rod is quadrupolar, driving liquid towards the rod along the electric field and forcing it away from the rod in the direction perpendicular to the imposed electric field. The measured ICEO flow velocity is proportional to the square of the electric field strength, and depends on the applied AC frequency.

3:20PM D6.00006 Nonlinear electrokinetic repulsion effects in combined electroosmotic and Poiseuille flow through microchannels[1] , NECMETTIN CEVHERI, MINAMI YODA, Georgia Institute of Technology — Recent evanescent-wave particle velocimetry studies in electrokinetically driven flow where aqueous solutions are driven by an electric field of magnitude E, have shown that the radius $a = O(0.1\text{-}1\ \mu\text{m})$ particle tracers suspended in the solution are subject to a wall-normal force that drives particles away from the wall [Kazoe & Yoda, *Langmuir* **27**:11481]. The magnitude of this force appears to scale as E^2 and a^2, albeit over a limited range of E and a, suggesting that particles of different sizes will have different average wall-normal positions, and hence sample different velocity distributions in a shear flow. To verify this hypothesis, evanescent-wave particle velocimetry was used to measure near-wall particle distributions and velocities of $a = 0.2\ \mu$m and 0.5 μm particles in the combined electroosmotic and Poiseuille flow of a bidisperse dilute aqueous solution through fused-silica channels about 30 μm deep for $E < 45$ V/cm and pressure gradients $\Delta p/L \leq 1.3$Bar/m. To evaluate the whether this nonlinear electrokinetic force can be used separate particles based on their size, near-wall particle distributions for both particle sizes were measured at different streamwise locations in the combined flow.

[1]Supported by ARO and NSF

3:33PM D6.00007 Electrokinetic investigations of uniformly dissociated polymer films , ALEXANDER BARBATI, BRIAN KIRBY, Sibley School of Mechanical and Aerospace Engineering, Cornell University — We execute electrokinetic investigations of a Nafion polymer film attached to a rigid glass substrate. These measurements reveal a film charging mechanism that follows Donnan potential scalings over several decades of hydronium and salt concentration, showing invariance of the film dissociation with respect to both pH and solution ionic strength. Electrokinetic measurements are additionally supplemented by observations characterizing the physical (ellipsometry) and chemical (XPS) film state. Our experimental results are analyzed using analytical and numerical modeling of the volumetrically–charged soft interface to interpret measured fluxes with more familiar quantities such as conductivities, surface potentials, and apparent slip lengths.

3:46PM D6.00008 Streaming Potential of an Electrolyte in a Microchannel with an Axial Temperature Gradient , MATHIAS DIETZEL, STEFFEN HARDT, Institute for Nano- & Microfluidics, Center of Smart Interfaces, TU Darmstadt — The effect of a temperature gradient parallel or antiparallel to the main flow direction of a pressure-driven symmetric electrolyte in a slit-microchannel is investigated. Based on the non-isothermal Nernst-Planck equations as well as on the Poisson equation, and under the assumption that the intrinsic Soret coefficient S^* is the same for each ion species, an analytical expression of the electric double layer (EDL) potential is derived. Since the local EDL thickness is found to increase exponentially with temperature, a temperature difference δT applied along a channel exhibiting a constant surface zeta potential leads to a corresponding gradient in the EDL thickness. For large pressure differences δp_0, the non-isothermal streaming potential can be adequately described by the well known isothermal expression if the local modification of the Debye length due to the thermal effect is taken into account. For small channel heights at small driving pressure differences, the streaming potential is seen to be over-predicted (under-predicted) by the (Debye-length corrected) isothermal expression for positive (negative) values of $S^*\delta T/\delta p_0$. With vanishing pressure difference, the steady-state thermoelectric potential of confined electrolytes is derived.

3:59PM D6.00009 Ion Altered Fluorescence Imaging (IAFI): A Non-invasive, Visualization Method Which Simultaneously Images Scalar Fields and Quantifies Local Ion Concentration[1] , VIKTOR SHKOLNIKOV, JUAN G. SANTIAGO, Stanford University — Electrokinetic flows are leveraged for a wide range of microfluidic and lab-on-a-chip systems, and are often used to mix, preconcentrate, and/or separate analytes. Traditionally, temperature, conductivity, electrochemical, and UV absorbance detectors have been used to indirectly estimate analyte concentration profiles in these flows. However, these typically are point detectors and thus do not permit dynamic, full-field visualization of unsteady scalar fields. To address this, we propose a novel visualization and quantitation method we term ion altered fluorescence imaging (IAFI). IAFI leverages florescence quenching or enhancement of electrically neutral dyes by ions. IAFI therefore provides a non-intrusive quantitation of full-field concentration of non-fluorescent ions endogenous to the flow and its application. We demonstrate this method in visualization of two non-linear electrokinetic flows: isotachophoresis (ITP) and electrokinetic instability (EKI) in an electrokinetic focusing flow. We have quantified shock propagation and ion concentrations upstream and downstream of shocks in cationic and anionic ITP. We quantified and visualized chaotic EKI flow, including complex secondary flows and local ion densities as the flow develops downstream.

[1]This work was supported by National Science Foundation (NSF) grant CBET-0967600-000. V.S. was supported by NSF GRF.

4:12PM D6.00010 Diffusion of molecules along incompressible interfaces due to electric fields , EBRAHIM KOLAHDOUZ, DAVID SALAC, University at Buffalo SUNY — The diffusion of insoluble molecules, such as surfactants or lipids, on incompressible interfaces due to electric fields is important in understanding the behavior of vesicles. Here a three-dimensional model is presented to investigate the motion of molecules on an arbitrary curved and incompressible interface in the presence of electric fields and an analytic fluid flow field. The interface is described using the gradient-augmented level set method while the electric field is solved with the immersed interface method. The motion of molecules on the surface are modeled using a set of coupled convection-diffusion equations. These equations are solved using the implicit closest point method. The model and sample results are presented.

Sunday, November 18, 2012 2:15PM - 4:25PM –

Session D7 Microfluidics: Methods and Devices II 24C - Chair: Manu Prakash, Stanford University

2:15PM D7.00001 Simultaneous measurement of the geometry and the internal 3D velocity field of a micron sized droplet confined in a channel using Astigmatism-PTV , TOBIAS MACK, CHRISTIAN CIERPKA, CHRISTIAN J. KÄHLER, Bundeswehr University Munich — Astigmatism-PTV is a method that allows to measure with a single camera the fully three-dimensional, three-component velocity field. The technique is ideally suited for microfluidic velocity measurements without errors due to in-plane and out-of-plane averaging (Cierpka et al. Meas Scie Tech 21, 2010). Recently it was shown, that the interface between two fluids or the surrounding fluid and droplets or bubbles can be estimated as well with the technique (Rossi et al., Meas Scie Tech 22, 2010). In this contribution the advantages of both techniques are combined to measure the shape of a droplet inside a micro channel along with the internal 3D flow field of the droplet induced by the surrounding fluid. For the current investigation, particles were only distributed within oil-droplets. Therefore the shape of the droplet could be later reconstructed by the volumetric particle positions and the velocity can be estimated tracking the same particles in consecutive frames of the same dataset. The procedure allows the simultaneous determination of the shape and the droplet velocity as well as the inner flow field and offers a great potential for current research.

2:28PM D7.00002 Correlation Force Spectroscopy for Single Molecule Measurements[1] , MILAD RADIOM, Chemical Engineering, Virginia Tech, BRIAN ROBBINS, Mechanical Engineering, Virginia Tech, JOHN WALZ, Chemical Engineering, Virginia Tech, MARK PAUL, Mechanical Engineering, Virginia Tech, WILLIAM DUCKER, Chemical Engineering, Virginia Tech — The sensitivity of force measurements in single molecule force spectroscopy is limited by noise resulting from thermal vibration of microcantilevers as well as hydrodynamic interaction with surrounding fluid. The thermal noise is particularly important in biomolecular studies since many biomolecular events have energy similar to the thermal noise: for a cantilever of spring constant k_s ~0.1N/m the thermal noise sets a bound on force noise level of order $(k_s k_B T)^{1/2}$ ~20pN where k_B and T are the Boltzmann constant and temperature. We describe a new technique which characterizes single polymer chains through measurement of correlations between thermally-stimulated vibrations of two closely spaced microcantilevers in fluid. We call this technique Correlation Force Spectrometry. CFS has lower noise than its counterpart single cantilever techniques since thermal noise in CFS only arises from the region where the two cantilevers interact due to fluid coupling, rather than from the entire cantilever. When a molecule is straddled between the two cantilevers, the correlation arises from the solvent as well as stiffness and internal damping of the molecule. We will present our results showing the effect of coupling through a single molecule of dextran.

[1]National Science Foundation Award Number CBET-0959228

2:41PM D7.00003 Micro-Particle Image Velocimetry using Microfabricated Diode Lasers , NICHOLAS JUDY, UC Santa Barbara — Microfabricated diode lasers are interfaced with PDMS microchannels using in-plane waveguides. This allows for micro-PIV measurements to be obtained in the microchannel, without the requirement of large external Nd:YAG lasers, which are commonly used in traditional micro-PIV. The microfabricated diode laser has a peak wavelength of 532nm and produces up to 200mW of power. PDMS waveguides are designed in-plane and perpendicular as a part of the PDMS microdevices and coupled to the microfabricated laser using an optical fiber. Two types of waveguides are designed: a PDMS waveguide and a microfluidic waveguide. The PDMS waveguide involves a two-step soft-lithography process to give a difference in index, while the microfluidic waveguide uses oil as the waveguide medium. Limitations of the current technique and its potential impact on the future of micro-PIV will be presented.

2:54PM D7.00004 ABSTRACT WITHDRAWN –

3:07PM D7.00005 Simple and inexpensive micro-capillary devices for generating composite emulsions , ERQIANG LI, JIAMING ZHANG, SIGURDUR THORODDSEN, King Abdullah University of Science and Technology, Saudi Arabia — All-glass microfluidic devices have attracted recent attention due to their excellent chemical robustness, bio-compatibility, optical properties and the ease of modifying their surface wettability. Herein we report the usage of a single tapered cylindrical glass capillary and microscope slides to fabricate simple and inexpensive all-glass microfluidic devices that are capable of producing monodisperse double emulsions. Triple emulsion droplets of water-in-oil-in-water-in-oil (W/O/W/O) or O/W/O/W phases can also be stably generated by adding another cylindrical capillary next to the outlet of the first capillary. In addition, by careful controlling the wettability of the inner surface of the first capillary, multi-component emulsion droplets of (gas and water)-in-oil-in-water ((G+W)/O/W) phases can also be stably produced. Such gas-laden emulsion drops may be beneficial for bio-related applications where oxygen supply is required. The relationship between the flow parameters and the resulting number of encapsulated droplets and the emulsion droplet sizes, have been investigated, for all of these various higher order emulsions.

3:20PM D7.00006 Pumpless Transport of Low Surface Tension Liquids in Surface Tension Confined (STC) Tracks , CONSTANTINE MEGARIDIS, THOMAS SCHUTZIUS, MOHAMED ELSHARKAWY, University of Illinois at Chicago, MANISH TIWARI, Swiss Federal Institute of Technology, ETH-Zurich — Surfaces with patterned wettability have potential applications in microfluidics, fog capture, pool boiling, etc. With recent fabrication advancements, surfaces with adjacent superhydrophobic and superhydrophilic regions are feasible at a reasonable cost; with properly designed patterns, one can produce microfluidic paths (a.k.a. surface tension confined or STC tracks) where a liquid is confined and transported by surface tension alone. The surface tension of water is relatively high (72 mN/m), as compared with oils (~25 mN/m) and organic solvents (~20 mN/m). This makes the design of STC channels for oils and organic solvents far more difficult. In this study, open STC tracks for pumpless transport of low-surface tension liquids (acetone, ethanol, and hexadecane) on microfluidic chips are fabricated using a large-area, wet-processing technique. Wettable, wax-based, submillimeter-wide tracks are applied by a fountain-pen procedure on superoleophobic, fluoroacrylic carbon nanofiber (CNF) composite coatings. The fabricated anisotropic wetting patterns confine the low-surface tension liquids onto the flow tracks, driving them with meniscus velocities exceeding 3 cm/s. Scaling arguments and Washburn's equation provide estimates of the liquid velocities measured in these tracks, which also act as rails for directional sliding control of mm-sized water droplets. The present facile patterned wettability approach can be extended to deposit micrometer-wide tracks.

3:33PM D7.00007 Selective pumping in a network: A novel bioinspired flow transport paradigm , YASSER ABOELKASSEM, ANNE STAPLES, Virginia Tech — We present a new paradigm for selectively pumping and controlling fluids at the microscale in a complex network of channels, which we call "selective pumping in a network." The approach is inspired by internal flow distributions induced by rhythmic wall contraction phenomena in insect tracheal networks. The selective pumping concept presented enables fluids to be transported, controlled and directed into specific branches in networks while avoiding other possible branching routes, without the use of any mechanical valves. The results presented here might help guide efforts to fabricate novel microfluidic devices with improved efficiency for mixing purposes and targeted drug delivery applications. In this study, both theoretical analysis and Stokeslets-meshfree computational methods are used to solve for the 2D viscous flow transport in an insect-like tracheal network of channels with prescribed moving wall contractions. The derived theoretical analysis is based on both lubrication theory and quasi-steady approximations at low Reynolds numbers. The meshfree numerical method is based on the method of fundamental solutions (MFS) that uses a set of singularized force elements "Stokeslets" to induce the flow motions. Moreover, the passive particle tracking simulation

3:46PM D7.00008 Hand-powered microfluidics: A membrane pump with a patient-to-chip syringe interface , BRENDAN MACDONALD, MAX GONG, University of Toronto, TRUNG NGUYEN, National Hospital for Tropical Diseases, Hanoi, Vietnam, DAVID SINTON, University of Toronto — In this talk, an on-chip hand-powered membrane pump with a robust patient-to-chip syringe interface is presented. This approach enables safe sample collection, sample containment, integrated sharps disposal, high sample volume capacity, and controlled downstream flow with no electrical power requirements. Sample is manually injected into the device via a syringe and needle. The membrane pump inflates upon injection and subsequently deflates, delivering fluid to downstream components in a controlled manner. The device is fabricated from poly(methyl methacrylate) (PMMA) and silicone, using CO_2 laser micromachining. Pump performance is experimentally demonstrated and the behavior is subsequently modeled with reference to a resistor-capacitor electrical circuit analogy. Downstream output of the membrane pump is regulated, and scaled, by connecting multiple pumps in parallel. The device provides precisely controlled pumping rates and high volume throughput without any electrical power requirements.

3:59PM D7.00009 Digitally controlled droplet microfluidic system based on electrophoretic actuation[1] , DO JIN IM, BYEONG SUN YOO, MYUNG MO AHN, DUSTIN MOON, IN SEOK KANG, Pohang University of Science and Technology (POSTECH) — Most researches on direct charging and the subsequent manipulation of a charged droplet were focused on an on-demand sorting in microchannel where carrier fluid transports droplets. Only recently, an individual actuation of a droplet without microchannel and carrier fluid was tried. However, in the previous work, the system size was too large and the actuation voltage was too high (1.5 kV), which limits the applicability of the technology to mobile use. Therefore, in the current research, we have developed a miniaturized digital microfluidic system based on the electrophoresis of a charged droplet (ECD). By using a pin header socket for an array of electrodes, much smaller microfluidic system can be made from simple fabrication process with low cost. A full two dimensional manipulation (0.4 cm/s) of a droplet (300 nL) suspended in silicone oil (6 cSt) and multiple droplet actuation have been performed with reasonable actuation voltage (300 V). By multiple droplet actuation and coalescence, a practical biochemical application also has been demonstrated. We hope the current droplet manipulation method (ECD) can be a good alternative or complimentary technology to the conventional ones and therefore contributes to the development of droplet microfluidics.

[1]This work has been supported by BK21 program of the Ministry of Education, Science and Technology (MEST) of Korea.

4:12PM D7.00010 Puch Card Programmable Microfluidics , GEORGE KORIR, MANU PRAKASH, Stanford University — Microfluidic technology has emerged as a powerful means of manipulating fluids at the micro-scale with many promising applications, but universal programmability is still dependent on external control systems. With a focus on global-health and field applications, external control and pumps significantly hamper the use of microfluidic devices in harsh conditions. Punch Cards (simple tapes of paper) have been used to program early computers before the advent of electronic memory. With this analogy, we present a novel universal programming scheme for microfluidics using paper Punch Card tapes. We further characterize our devices as a function of readout speed, bit-error rate for a given operating conditions (Capillary and Reynolds number). A lumped element model was built to characterize the flow and serve as a predictive template for future designs. Operated by hand, the system requires no external sources of electricity or pumps. We demonstrate that Punch Cards provide an innumerable number of ways to program fluids including running complex protocols in the field with minimal training.

Sunday, November 18, 2012 2:15PM - 4:25PM –

Session D8 Particles: Experimental 25A - Chair: Greg Voth, Wesleyan University

2:15PM D8.00001 In situ calibration of volume concentration measurements using PTV correlation for particle-laden flows , RAHUL MULINTI, KYLE CORFMAN, KEN KIGER, University of Maryland — Determination of the effective measurement volume is essential for making quantitative concentration measurements of a dispersed phase when using particle-imaging technique for two-phase flows. While nominally determined by the local light sheet thickness, the actual value depends also on the dispersed phase identification characteristics used to detect the particles (relative brightness, size, etc.), and stray illumination such as scattering by tracer particles and wall reflections, necessitating use of local calibration techniques. In the current work, a novel *in situ* method is proposed where the effective light sheet thickness is estimated using particle image correlation information of free falling dispersed-phase particles settling through a tilted light sheet. Increasing the delay time between the image pairs results in in-plane loss of correlation as the particle images move out of the light sheet. By employing a threshold on the height of the normalized mean correlation peak, and relating this to the actual particle image identification characteristics, the effective light sheet thickness is estimated. The effects of tracer particles and presence of a strong wall reflection on the effective light sheet thickness is also reported.

2:28PM D8.00002 Using direct numerical simulation to improve experimental measurements of inertial particle radial relative velocities , PETER J. IRELAND, LANCE R. COLLINS, Cornell University — Turbulence-induced collision of inertial particles may contribute to the rapid onset of precipitation in warm cumulus clouds. The particle collision frequency is determined from two parameters: the radial distribution function $g(r)$ and the mean inward radial relative velocity $\langle w_r^{(-)} \rangle$. These quantities have been measured in three dimensions computationally, using direct numerical simulation (DNS), and experimentally, using digital holographic particle image velocimetry (DHPIV). While good quantitative agreement has been attained between computational and experimental measures of $g(r)$ (Salazar et al. 2008), measures of w_r have not reached that stage (de Jong et al. 2010). We apply DNS to mimic the experimental image analysis used in the relative velocity measurement. To account for experimental errors, we add noise to the particle positions and 'measure' the velocity from these positions. Our DNS shows that the experimental errors are inherent to the DHPIV setup, and so we explore an alternate approach, in which velocities are measured along thin two-dimensional planes using standard PIV. We show that this technique better recovers the correct radial relative velocity PDFs and suggest optimal parameter ranges for the experimental measurements.

2:41PM D8.00003 Rotation in turbulence of aquatic organisms modeled as particles , EVAN VARIANO, MARGARET BYRON, GABRIELE BELLANI, UC Berkeley — We investigate which length and time scales are relevant for determining the rotation of aquatic organisms and their gametes. We are interested in parameter space beyond the Stokes regime, and also the effect of particle shape on rotation. We report experimental measurements that use custom–manufactured particles to model aquatic organisms, which are designed with the necessary optical properties so that we can measure their rotation, simultaneously with the vorticity statistics of the surrounding fluid. Lagrangian timeseries of particles' angular velocity allows investigation of rotational diffusion.

2:54PM D8.00004 Turbulence modulation by Taylor-scale particles: in search of a universal parameter , GABRIELE BELLANI, JEFF SEMIGRAN, MARGARET BYRON, EVAN VARIANO, University of California Berkeley — In this work we investigate turbulence modulation effects by Taylor microscale-sized particles in homogeneous isotropic turbulence. We present a novel experimental technique that allows us to explore a wide range of parameters in terms of particle shape and concentration. In lab experiments we perform Particle Image Velocimetry on both the fluid phase and cross-sections of suspended particles. These measurements yield the fluid-phase velocity, linear and angular velocities of the particles, as well as the detail of the flow near the particle interface. We report turbulence modulation effects as function of particle shape and concentration. These include change in turbulent kinetic energy, mean dissipation rates, and spectral analysis. The results are normalized using various scaling factors, in an attempt to find a universal parameter that describes turbulence modulation for particles of any shape and size. The details of the interphase-coupling mechanisms are also investigated by comparing statistics of particle translation and rotation rates to statistics of the fluid-phase velocity field at various length scales.

3:07PM D8.00005 Particle behavior in linear shear flow: an experimental and numerical study[1] , NIMA FATHI, The University of New Mexico, MARC INGBER, University of Colorado - Denver, PETER VOROBIEFF, The University of New Mexico — We study particle behavior in low Reynolds number flows. Our experimental setup can produce both Couette flow and Pouseuille flow at low Reynolds numbers. Spherical particles are suspended in gravity-stratified Newtonian fluid. Their predominantly two-dimensional motion is driven by moving belts (and/or piston) that produce shear in the fluids. Particle migration and translational velocity have been studied. The irreversibility of particle motion has been investigated. The experimental results are compared to the numerical simulations performed with discrete phase element method (DPM). Particle trajectories with the same boundary conditions in viscous fluids have been studied. The irreversibility in numerical simulation has been modeled for different cases. Results show the particle migration is a function of shear rate, particle size, degree of symmetry of the fluid domain, and also of the initial starting position, the latter playing an important role in the irreversibility of particle motion.

[1]This research is partly supported by Procter & Gamble.

3:20PM D8.00006 Measuring the effects of finite length of rods on rotation rate in turbulent flow , SHIMA PARSA, GREG VOTH, Wesleyan University — We study the rotational dynamics of single rod-like particles ranging from tracer rods to long rods and quantify the effects of length of rod on it's rotation rate in turbulent flow. The position and orientation of rods are measured experimentally using Lagrangian particle tracking with multiple cameras in a flow between oscillating grids.
The rotation rate of rods is determined by the velocity gradient of the flow. As rods are transported in the flow they develop alignment with the directions of the vorticity and eigenvectors of the strain-rate. This alignment results in a smaller rotation rate from randomly oriented rods. Tracer rods rotate due the velocity gradient of the smallest eddies that produce the largest shear rate while longer rods average over length-scales ranging from eddies smaller than their size to eddies order of their own length-scale. As the length of the rods increases the rotation rate variance gets smaller.

3:33PM D8.00007 Lagrangian Measurements of Vorticity and the Rotational Dyanmics of Anisotropic Particles in Turbulence , GUY GEYER, SHIMA PARSA, STEFAN KRAMEL, GREG VOTH, Wesleyan University — We measure the Lagrangian rotational dynamics of anisotropic particles in turbulent flow using stereoscopic video imaging. Using 3D printing technology, we fabricate rods, crosses (two perpendicular rods), and jacks (three mutually perpendicular rods). The three dimensional position and orientation of these objects can be reconstructed using a combination of stereomatching and optical tomography. We apply these techniques to measurements in a $R_\lambda \approx 200$ flow, where turbulence is generated by two grids oscillating in phase. Since the advected particles have a largest dimension less than 10 times the Kolmogorov length, they are good approximations of tracer particles. Using resistive force theory, we demonstrate that tracer jacks and crosses have the same rotational dynamics as spheres and disks, respectively. Thus, we can measure the rotation rates of ellipsoidal particles at aspect ratios, α, that span the entire range: $\alpha \approx 0$ (disks) $\alpha = 1$ (spheres), and $\alpha \approx \infty$ (rods). Furthermore, measurements of the rotation rate of jacks constitute a novel method for obtaining Lagrangian measurements of the vorticity in turbulent flows.

3:46PM D8.00008 ABSTRACT WITHDRAWN –

3:59PM D8.00009 Simultaneous 3D measurement of the translation and rotation of finite size particles and the flow field in a fully developed turbulent water flow[1] , MATHIEU GIBERT, SIMON KLEIN, EBERHARD BODENSCHATZ, MPI-DS — We report a novel experimental technique that measures simultaneously in three dimensions the trajectories, the translation, and the rotation of finite size inertial particles together with the turbulent flow. The flow field is analyzed by tracking the temporal evolution of small fluorescent tracer particles. The inertial particles consist of a super-absorbent polymer that renders them index and density matched with water and thus invisible. The particles are marked by inserting at various locations tracer particles into the polymer. Translation and rotation, as well as the flow field around the particle are recovered dynamically from the analysis of the marker and tracer particle trajectories. We apply this technique to study the dynamics of inertial particles much larger in size ($Rp/\eta \approx 100$) than the Kolmogorov length scale η in a von Kármán swirling water flow ($R_\lambda \approx 400$). We show, using the mixed (particle/fluid) Eulerian second order velocity structure function, that the interaction zone between the particle and the flow develops in a spherical shell of width 2Rp around the particle of radius Rp. This we interpret as an indication of a wake induced by the particle. (http://arxiv.org/abs/1205.2181)

[1]This work was funded generously by the Max Planck Society and the Marie Curie Fellowship, Program PEOPLE - Call FP7-PEOPLE-IEF-2008 Proposal No 237521. Support from COST Action MP0806 is kindly acknowledged.

4:12PM D8.00010 Hydrodynamic alignment of Nano-Fibrillated Cellulose in extensional flow , KARL HÅKANSSON, FREDRIK LUNDELL, LISA PRAHL WITTBERG, Linne FLOW Centre, KTH Mechanics, Royal Institute of Technology, SE-10044 Stockholm, Sweden, LARS WÅGBERG, Fibre and Polymer Technology, Royal Institute of Technology, SE-10044 Stockholm, Sweden, DANIEL SÖDERBERG, Innventia AB, Box 5604, SE-11486 Stockholm, Sweden, WALLENBERG WOOD SCIENCE CENTER TEAM — The aim of this work is to manipulate the orientation of cellulose fibrils in order to enable control of material properties. Cellulose fibrils are the load bearing component of wood and separation of the fibrils from the cell wall is possible through enzymatic and mechanical treatment. The resulting product is called Nano-Fibrillated Cellulose, NFC, consisting of elongated particles with diameters of 40 nm and lengths of a few μm. Films and fibers made by NFC show great potential in terms of material properties. This work includes experiments, computations and simulations in order to determine the alignment of NFC in a laminar extensional flow. A flow focusing setup is used where water is accelerating a semi-dilute NFC-dispersion, this particular design minimizes the shear on the NFC-dispersion. The relative mean orientation is found through 2D birefringence measurements. The Smoluchowski equation including an orientational diffusion term and a flow forcing term is solved numerically in 1D. Flow field simulations are made in order to find the local acceleration, and also to confirm the shape of the suspension thread formed. The computations predict the same trend as is seen in the experiments; at higher accelerations the NFC-fibrils become more aligned.

Sunday, November 18, 2012 2:15PM - 4:25PM –

Session D9 Interfacial/Thin Film Instability II: Fingering 25B - Chair: Sarah Kieweg, University of Kansas

2:15PM D9.00001 Viscous fingering with production of surfactant by chemical reaction in a Hele-Shaw cell , MASANARI FUJIMURA, YUICHIRO NAGASTU, Tokyo University of Agriculture and Technology — Viscous fingering experiments have been performed in a radial Hele-Shaw cell for a liquid–liquid system in the presence of a chemical reaction which produces a surfactant. The reaction is a neutralization of a fatty acid by an alkaline material to form a surfactant. Viscous fingering experiments employing the chemical recipe were previously performed by several researchers. The present experiments were done in wider range of the reactant concentrations and the flow rate. Experimental results showed that the reaction made viscous fingers thinner for low flow rate whereas wider for high flow rate in the condition of low reactant concentrations. The reaction made the fingers wider for low reactant concentrations whereas thinner for high reactant concentration in the condition of high flow rate. In summary, we have found the opposite effects of the reaction on the finger width depending on flow rate in the low reactant concentration and depending on reactant concentrations in the high flow rate by employing the wide range of experimental conditions.

2:28PM D9.00002 Viscous fingering involving disappearance of precipitation by a chemical reaction in a Hele-Shaw cell , YUKI ISHII, YUTAKA TADA, Nagoya Institute of Technology, YUICHIRO NAGATSU, Tokyo University of Agriculture and Technology — Previously, we experimentally studied viscous fingering involving production of a precipitation by a chemical reaction in a Hele-Shaw cell (Nagatsu et al. PRE 77, 067302 (2008)). In the present study, we have conducted experiments on viscous fingering involving disappearance of a precipitation by a chemical reaction in a Hele-Shaw cell. In the present experiments, the more-viscous liquid contains the precipitation. In the reactive case, we used a solution including a reactant which reacts with the precipitation resulting in disappearance of the precipitation. In the non-reactive case, water was used as the less-viscous liquid. Thus, viscous fingering was observed in both the reactive and non-reactive cases. We have found that viscous fingering pattern is changed by disappearance of the precipitation by the reaction. Furthermore, effects of the reactant concentration and the injection rate of the less-viscous liquid on the change in the pattern by the disappearance of the precipitation were examined.

2:41PM D9.00003 Experimental Investigation of the Growth of Mixing Zone in Miscible Viscous Fingering , SAHIL MALHOTRA, ERIC R. LEHMAN, MUKUL M. SHARMA, The University of Texas at Austin — An experimental study is performed to study the growth of the mixing zone in miscible viscous fingering. Rectilinear flow displacement experiments are performed in a Hele-Shaw cell over a wide range of viscosity ratios (1 to 700) by injecting water into Glycerol solutions at different flow rates. A linear growth in mixing zone is observed in all the experiments. The mixing zone velocity increases with the viscosity ratio up to viscosity ratios of 330 and the trend is consistent with Koval's model (Koval 1963). However, at higher viscosity ratios the mixing velocity plateaus signifying no further effect of viscosity contrast on the growth of mixing zone.

2:54PM D9.00004 ABSTRACT WITHDRAWN –

3:07PM D9.00005 Anomalous structure formation in the zero surface tension limit of viscous fingering , IRMGARD BISCHOFBERGER, RADHA RAMACHANDRAN, SIDNEY R. NAGEL, The University of Chicago, Department of Physics, THE UNIVERSITY OF CHICAGO TEAM — The displacement of a more viscous fluid, of viscosity η_{out}, by a less viscous one, of viscosity η_{in}, in a two-dimensional geometry or a porous medium is unstable and typically produces complex fingering patterns. These fingering patterns are predicted to become sharper as the surface tension between the two fluids is decreased. However, our experiments performed in a radial Hele-Shaw cell suggest the opposite conclusion: fingering is less likely to occur in the limit of low surface tension. When the two fluids are miscible, so that the surface tension is negligible, the instability can be entirely suppressed; when the viscosity ratio of the two fluids, η_{out}/η_{in}, is greater than, but close to, one, the interface between the fluids is circular. With increasing viscosity ratio, the pattern starts to develop small blunt structures (toes) and when the viscosity ratio is large the pattern consists of highly branched fingers. We measure the amount of external fluid that gets displaced by the less viscous one and find that the displacement across the gap is always incomplete; the fingers form three dimensional structures. We discuss the implications of this 3D nature of the instability on the overall pattern formation.

3:20PM D9.00006 Fingering instabilities for a thin liquid film flowing down the outside of a vertical cylinder[1] , SCOTT MCCUE, LISA MAYO, TIMOTHY MORONEY, Queensland University of Technology — The flow of a thin film of viscous fluid down an inclined plane is well-studied, with much progress made by applying the lubrication approximation to derive a governing evolution equation for the film height. This equation is a fourth-order pde with a nonlinear degenerate diffusion term. Here we generalise this approach to apply for the problem of a thin film flowing down the outside of a vertical cylinder. In this context, a recent linear stability analysis of Smolka & SeGall [1] provides a relationship between the growth rate and wavenumber of each mode, predicting the number of fingers that form on the surface of the cylinder as a function of the fluid properties and the cylinder's radius. To complement these results, we solve the full nonlinear problem numerically and analyse the manner in which nonlinear modes grow and interact for longer times. We also consider the problem of a single large droplet spreading and sliding down the vertical cylinder, studying the effect that the cylinder curvature has on the flow.

[1] L.B. Smolka and M. SeGall, "Fingering instability down the outside of a vertical cylinder," Phys. Fluids **23**, 092103 (2011)

[1]We acknowledge support from the ARC Linkage Project LP100200476.

3:33PM D9.00007 Contact line instability of gravity-driven flow of power-law fluids , BIN HU, SARAH KIEWEG, University of Kansas — In our previous studies, we developed 2D and 3D models to simulate a power-law fluid flowing down an incline. This study is intended to examine how the shear-thinning effect of the fluid can influence the fingering instability for arbitrary wavenumbers in gravity-driven thin film flow. We apply the linear stability analysis method on our 3D power-law model and use Taylor series to approximate the power terms in the power-law evolution equation. The perturbation and the growth rate are obtained numerically for different wavenumbers. Parametric study is performed to investigate the impact of shear-thinning index on the growth rate of perturbation. For the assessment of this study, we compare the result of this study with the existing result for Newtonian fluids in literature. The wavelength and growth rate obtained in this study are also compared to our previous 3D simulation results and experimental results.

3:46PM D9.00008 Inhibition of viscous fluid fingering: A variational scheme for optimal flow rates[1] , JOSE MIRANDA, EDUARDO DIAS, Depto. de Fisica - UFPE, BRAZIL, ENRIQUE ALVAREZ-LACALLE, Dept. Fisica Aplicada, UPC, SPAIN, MARCIO CARVALHO, Depto. Eng. Mec., PUC-Rio, BRAZIL — Conventional viscous fingering flow in radial Hele-Shaw cells employs a constant injection rate, resulting in the emergence of branched interfacial shapes. The search for mechanisms to prevent the development of these bifurcated morphologies is relevant to a number of areas in science and technology. A challenging problem is how best to choose the pumping rate in order to restrain growth of interfacial amplitudes. We use an analytical variational scheme to look for the precise functional form of such an optimal flow rate. We find it increases linearly with time in a specific manner so that interface disturbances are minimized. Experiments and nonlinear numerical simulations support the effectiveness of this particularly simple, but not at all obvious, pattern controlling process.

[1]J.A.M., E.O.D. and M.S.C. thank CNPq/Brazil for financial support. E.A.L. acknowledges support from Secretaria de Estado de IDI Spain under project FIS2011-28820-C02-01

3:59PM D9.00009 Modelling the suppression of radial fingering in elastic Hele-Shaw cells , DRAGA PIHLER-PUZOVIC, University of Manchester, RAPHAEL PERILLAT, University of Manchester, Ecole Centrale de Lyon, MATTHIAS HEIL, ANNE JUEL, University of Manchester — We find a surprisingly effective means of suppressing the fingering instabilities at the interface of air and a viscous fluid in the narrow gap between two parallel plates, by replacing one of the plates with an elastic membrane. Experiments show that the resulting fluid-structure interaction considerably delays the onset of fingering and fundamentally alters the large-amplitude interfacial patterns that develop subsequently. We present the results of a linear stability analysis which assesses how the stability of the axisymmetrically expanding air bubble to non-axisymmetric perturbations is affected by the presence of the elastic membrane, and perform direct numerical simulations to follow the evolution of the instabilities into the large-amplitude regime. The theoretical/computational predictions are then compared against the experimental results.

4:12PM D9.00010 Thin-film flows without precursors , RUBEN JUANES, LUIS CUETO-FELGUEROSO, MICHAEL SZULCZEWSKI, MIT — The flow of thin films over flat surfaces has been the subject of much theoretical, experimental and computational research. Under 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. This classical model, however, effectively assumes that the film is perfectly wetting to the substrate and, therefore, does not capture the partial wetting regime. Partial wetting is responsible for stopping the spread of a liquid puddle, and for pinning the contact line of a viscous liquid down an incline, controlling the morphology of the fingering pattern that ensues. Here, we extend our recent work on macroscopic phase-field modeling of two-phase flow in a capillary tube to thin-film flows with partial wetting. Our model naturally accounts for the dynamic contact angle at the contact line, and therefore permits modeling thin-film flows without invoking a precursor film, leading to compactly-supported solutions. We model the statics and dynamics of a liquid puddle, and the fingering behavior of flow down an incline. We compare model predictions with experiments of thin-film flows both on a horizontal plane and down an incline, for different contact angles.

Sunday, November 18, 2012 2:15PM - 3:59PM –

Session D10 Instability: Jets, Wakes and Shear Layers II: Wakes I 25C - Chair: Alis Ekmekci, University of Toronto

2:15PM D10.00001 The response of the wake past a bullet-shaped body to axisymmetric ZNMF forcing at high Reynolds numbers , GEORGIOS RIGAS, AIMEE MORGANS, JONATHAN MORRISON, Imperial College London — It has been generally acknowledged that wakes of axisymmetric bodies are dominated by the shedding of large-scale coherent structures of azimuthal wavenumbers $m = \pm 1$. In the present study, the effect of harmonic axisymmetric forcing ($m = 0$) on the vortex shedding mode is investigated experimentally for $Re = 2 \cdot 10^5$. The shedding mode was found to be insensitive to low levels of axisymmetric excitation, whereas for moderate levels a strong parametric subharmonic (2:1) resonance occurs among them. We show that the predictions of a model based on two coupled Stuart-Landau equations describing the evolution of the vortex shedding mode, nonlinearly coupled with the forced axisymmetric mode, are in good agreement with the experimental findings. The model parameters are identified based on experimental data.

2:28PM D10.00002 Modification of mean wake flow behind very slender axially symmetric bodies by nonlinear convectively unstable helical modes , J.T.C. LIU, KISEOK LEE, School of Engineering, Brown University — Recent experiments of Asai, et al. (2011) confirm earlier experiments of Sato & Okada (1966), Peterson & Hama (1976) that, for sufficiently slender axially symmetric bodies placed in a stream parallel to the axes, only convectively unstable modes exist. This is exploited theoretically (and computationally) by imposition of the most unstable helical modes to modify the otherwise round laminar wake flow. The local linear region is first considered theoretically to obtain the group velocities as a function of the streamwise distance, and which compared well with existing measurements. This information in used as convection velocity in a time-dependent nonlinear computation, as suggested by Spalart & Yang (1987) in the boundary layer case. The Reynolds stress modification of the developing laminar mean wake flow is assessed. The round wake is modified into an elliptic-like cross section for equal amplitudes for the n=+1,-1 modes; the consequences of unequal upstream amplitudes, such as a would be found in a slight axis misalignment, are presented. Accompanied are the energy transfer mechanisms between the mean flow and the modal content and that between the modes.

2:41PM D10.00003 Transitions to chaos in the wake of an axisymmetric bluff body[1] , YANNICK BURY, THIERRY JARDIN, ISAE, DAEP TEAM — This work aims at understanding the dynamical process that leads to the onset of chaos in the flow past a blunt-based axisymmetric bluff body. On the basis of direct numerical simulations, conducted for Reynolds numbers ranging from 100 to 900, we show that the flow undergoes multiple transitions, successively giving rise to the SS, RSPa, RSPb, RSPc and RSB wake states. In particular, the RSPc state, revealed in this work via long-term computations, is characterized by intermittent vortex stretching denoting the onset of chaos and the potential occurence of a third instability that superimposes to the first and second instability associated with state RSPa and RSPb respectively. Interestingly, the reflectional symmetry that characterizes the RSP states is still retained. Hence, chaos is triggered before the symmetry breaking and the occurence of the RSB state.

[1]The authors acknowledge the French Ministry of Defence and DGA for funding this work.

2:54PM D10.00004 The turbulent wake of a submarine model at varying pitch and yaw angle , ANAND ASHOK, ALEXANDER SMITS, Princeton 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 submarine model (DARPA SUBOFF model). Measurements in the wake were performed at a Reynolds number based on the length of 2.4×10^6. Mean velocity and three-component turbulence measurements were performed using Pitot probes and cross wires in the span-wise plane at three different downstream positions: 5, 7.5 and 10 diameters downstream of the trailing edge. The pitch and yaw angles were in the range 0 and $\pm 10^\circ$. Work supported by ONR Grant N00014-09-1-0263.

3:07PM D10.00005 On the Control of Flow past a Circular Cylinder: Use of a Singe Spanwise Protrusion , ALIS EKMEKCI, TAYFUN AYDIN, ANTRIX JOSHI, University of Toronto — An experimental investigation is conducted to evaluate the effects of a single spanwise protrusion on the flow past a circular cylinder. Consideration is given to a range of Reynolds numbers from 5,000 to 30,000 and three different protrusions of circular cross-section that vary in diameter from 2.9% to 5.9% of the cylinder diameter. Varying the angular location of the protrusion on the cylinder surface, critical locations are investigated via hot-film anemometry and hydrogen-bubble visualization. For all the Reynolds numbers and protrusion sizes studied, two angular locations are shown to be the most critical. These angles depend on the wire size and Reynolds number. Protrusion at the first critical angle results in significant attenuation in the spectral amplitude of velocity fluctuations; whereas, at the second critical angle, it leads to amplification. Long-time records of hydrogen-bubble images show, at the first critical angle, recurrent appearance of periods with no detectable Karman vortex shedding and short periods during which regular Karman shedding resumes. Time traces of velocity fluctuations, obtained from hot-film measurements, also depict irregularities at this critical location.

3:20PM D10.00006 POD Analysis of the Wake Behind a Foamed and a Finned Cylinder , MORTEZA KHASHEHCHI, KAMEL HOOMAN, The University of Queensland, THOMAS ROESGEN, ETH Zurich, ANDREW OOI, The University of Melbourne, QGECE COLLABORATION, WALTER BASSET LABORATORY COLLABORATION, INST. OF FLUID DYNAMICS COLLABORATION — Particle Image Velocimetry (PIV) has been carried out to investigate the wake region behind a foamed and a finned cylinder. The experiments are conducted for a wide range of Reynolds numbers from 1000 to 10000. Two dimensional results of planar PIV reveal the important aspects of the local flow features of the circular finned and foamed cylinders. These include turbulent boundary layer development over the surface and a delayed separation of the flow resulting in a smaller wake size in each case. 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 all cases shows that the first two spatial modes are contained 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 three different cylinder types, the first four eigenmodes of the flow field were calculated and their structures were analyzed. The first four eigenmodes reveal the details about the global mean flow structure, with the large-scale structure being mainly related to the most energetic flow motion.

3:33PM D10.00007 Experimental sensitivity analysis of the global mode frequency of cylindrical bodies with blunt trailing edges at large Re , OLIVIER CADOT, ENSTA-ParisTech, MATHIEU GRANDEMANGE, ENSTA-ParisTech/ PSA Peugeot Citroën, VLADIMIR PAREZANOVIC, ENSTA-ParisTech, ENSTA TEAM — The global mode frequency modification of a square and a "D" shape cylinders due to the insertion in the flow of a small local and steady disturbance is investigated experimentally at Re=20000. Sensitivity maps are built by measuring the global mode frequency of the flow for many positions of the disturbance around the cylinder. Sensitive regions of either large or low frequencies are identified. It is shown that their spatial structures become independent on the size of the disturbance if this size is smaller to any boundary layers in the flow. In that case, the frequency changes are mostly interpreted by the ability of the disturbance to modify the mean flow through local vorticity injection or change in the local turbulent properties. Theoretical predictions of these sensitive regions from Meliga et al. will be presented in another abstract.

3:46PM D10.00008 Experimental investigation of flow instabilities behind a cube , LUKASZ KLOTZ[1], Warsaw University of Technology, Warsaw - Poland, SOPHIE GOUJON-DURAND, JOSÉ EDUARDO WESFREID, PMMH (ESPCI-CNRS) Paris, France — The wake behind a cube has been experimentally investigated in a water tunnel using LIF visualization and PIV method. Measurements were carried out for a moderate Reynolds numbers, ranging up to 400. The basic flow shows four pairs of trailing vortex. Subsequent regimes were detected, with regular and instationnary instabilities. Values of onsets of instability have been determined and the nonlinear evolution of the perturbations, discussed in the framework of Landau models. The streamwise vorticity, obtained from PIV measurements, has been decomposed into azimuthally Fourier modes. The obtained bifurcation branches show symmetry breaking, corresponding to each regime obtained. The experimental results we present are in good agreement with a previous numerical simulation. The observed sequence of transitions, for flow instabilities behind a cube, are compared with those reported for flow behind a sphere and disks.

[1]also at PMMH (ESPCI-CNRS) Paris, France

Sunday, November 18, 2012 2:15PM - 4:25PM –

Session D11 Microfluidics: Mixing 26A - Chair: Nadine Aubry, Carnegie Mellon University

2:15PM D11.00001 A Mapping method for mixing with diffusion , CONOR P. SCHLICK, IVAN C. CHRISTOV[1], PAUL B. UMBANHOWAR, JULIO M. OTTINO, RICHARD M. LUEPTOW, Northwestern University — We present an accurate and efficient computational method for solving the advection-diffusion equation in time-periodic chaotic flows. The method uses operator splitting which allows advection and diffusion steps to be treated independently. Taking advantage of flow periodicity, the advection step is solved with a mapping method, and diffusion is added discretely after each iteration of the advection map. This approach allows for a "composite" mapping matrix to be constructed for an entire period of a chaotic advection-diffusion process, which provides a natural approach to the spectral analysis of mixing. To test the approach, we consider the two-dimensional time-periodic sine flow. When compared to the exact solution for this simple velocity field, the operator splitting method exhibits qualitative agreement (overall concentration structure) for large time steps and is quantitatively accurate (average and maximum error) for small time steps. We extend the operator splitting approach to three-dimensional chaotic flows. Funded by NSF Grant CMMI-1000469.

[1]Present affiliation: Princeton University. Supported by NSF Grant DMS-1104047.

2:28PM D11.00002 Advective-diffusive mixing in microchannels , OLEKSANDR GORODETSKYI, MICHEL F.M. SPEETJENS, PATRICK D. ANDERSON, Eindhoven University of Technology, Netherlands — Laminar mixing is key to many processes in micro-devices and the mapping method is a proven and efficient approach to simulate and investigate mixing phenomena in realistic 3D geometries. However, the conventional mapping method is restricted to purely advective transport, while molecular diffusion is often significant, in particular in small-scale devices. A recent extension of the mapping method by Gorodetskyi et al. (Phys. Fluids, 2012, in press) allows inclusion of diffusion and thus greatly expands the application area of the technique. Diffusive mapping method enables in-depth analysis of the interplay between chaotic advection and diffusion in realistic systems by essentially the same procedure as successfully employed before for the purely convective approach. As a benchmark model for a realistic prototype system: the 3D staggered herringbone micro mixer is considered. Advective-diffusive mixing for various protocols of the mixer, that posses different dynamical properties, is investigated by means of the diffusive mapping method.

2:41PM D11.00003 Effect of viscosity contrast on mixing and dispersion in a capillary tube , AMIR A. PAHLAVAN, Mechanical Engineering, MIT, BIRENDRA JHA, LUIS CUETO-FELGUEROSO, RUBEN JUANES, Civil and Environmental Engineering, MIT, GARETH H. MCKINLEY, Mechanical Engineering, MIT — Microfluidic mixing has received a renewed attention during the past decade due to its ubiquitous presence in nature and novel industrial devices. Most microfluidic devices however operate in the Stokes flow regime, meaning that turbulence and inertia do not play any role in the mixing process. While many fundamental aspects of microfluidic mixing are now understood, and a variety of methods have been proposed to enhance mixing at low Reynolds number flows, the influence of viscosity contrast on the non-equilibrium physics of mixing remains to be explored. In this work we address this problem through numerical simulations and reduced-order modeling. We investigate the role of viscosity contrast on hydrodynamic instabilities that control the dispersion and mixing of miscible fluid flows in a capillary tube, and exploit this new understanding to propose strategies for enhancing mixing at the microscale.

2:54PM D11.00004 Chaotic fluid mixing by alternating microparticle topologies to enhance biochemical reactions , YANG GAO, ALEXANDER VAN REENEN, MARTIEN HULSEN, ARTHUR DE JONG, MENNO PRINS, JAAP DEN TOONDER, Eindhoven University of Technology — We report experimental results on chaotic mass transport induced by alternating topological changes of magnetic particle chains actuated by a rotating magnetic field. Results on the induced fluid flows, through particle tracing and mixing experiments, are obtained for (1) the regime of rigid chain rotation and (2) the regime wherein chains periodically fragment and reform. In the case of rigid rotating chains, the overall tracer particle trajectories are steady circles around the center of the microparticle chains. In the regime of periodic chain breaking and reformation, the tracer particle trajectories become chaotic. The level of mixing is measured utilizing a mixing index (M) in a water-dye system, i.e. in a perfectly mixed system M=0 while in an unmixed system M=1. When particle chains periodically break and reform, we observe that M decreases from 1 to 0.1 within 15 rotational cycles. We also report the effects of the different mixing regimes on a biological (streptavidin-biotin) binding reaction in the solution. We conclude that the alternating topological change of microparticle chains is an effective mechanism to achieve chaotic mixing and thereby promote and homogenize reactions in lab-on-a-chip systems.

3:07PM D11.00005 Mixing Diagnostics in Confined, High-Speed Droplet Collisions , BRIAN CARROLL, CARLOS HIDROVO, University of Texas at Austin — Fast mixing remains a major challenge in droplet-based microfluidics. The low Reynolds number operating regime of most mixing devices signifies orderly flows that are devoid of any inertial characteristics. To increase droplet mixing rates, a novel technique is under development that uses a high Reynolds number gaseous phase for droplet generation and transport and promotes mixing through binary droplet collisions at velocities near 1m/s. Limitations in existing mixing diagnostic methodologies has persuaded cultivation of a new technique for measuring droplet collision mixing in confined microchannels. The technique employs single fluorophore laser-induced fluorescence, custom image processing, and meaningful statistical analysis for monitoring and quantifying mixing in high-speed droplet collisions. Mixing progress is revealed through two statistics that separate the roles of convective rearrangement and molecular diffusion during the mixing process. The end result is a viewing window into the rich dynamics of droplet collisions with spatial and temporal resolutions of 1μm and 25μs, respectively. Experimental results obtained across a decade of Reynolds and Peclet numbers reveal a direct link between droplet mixing time and the collision convective timescale. This work provides valuable insight into the emerging field of two-phase gas-liquid microfluidics and opens the door to fundamental research possibilities not offered by traditional oil-based architectures.

3:20PM D11.00006 3D mixing near the surface of actuated beads[1] , NEEHAR MOHARANA, MICHEL SPEETJENS, RUBEN TRIELING, HERMAN CLERCX, Eindhoven University of Technology, The Netherlands — Mixing in microfluidic devices is often a challenge because of the absence of turbulence. Here we must instead resort to laminar mixing by chaotic advection. In order to achieve chaotic advection in such devices, we explore a promising future technology for active mixing by actuating microscopic magnetic beads with magnetic fields. The present study addresses the fundamental transport phenomena and associated mixing processes around a piecewise-steadily translating and/or rotating spherical bead. A detailed transport analysis revealed that perturbation of the Stokes flow around the sphere is essential to attain (locally) chaotic mixing. To this end we introduce a nonlinear perturbation that changes the flow in essentially the same way as fluid inertia or bead oscillations. The impact of this perturbation on the mixing properties has been explored for various actuation protocols via symmetry analysis and numerical simulation of three-dimensional (3D) fluid trajectories and computation of Poincaré maps ("stroboscopic maps" of particle positions). We found evidence of intricate coherent structures which are key to 3D mixing in that they geometrically constrain and determine the tracer transport. Parallel to the above study, we have designed and manufactured a macro-scale experiment in order to validate the mixing properties around the sphere by 3D velocity and temperature measurements. Preliminary experimental results will be shown.

[1]This work is supported by the Dutch Technology Foundation STW under grant 10458.

3:33PM D11.00007 An Experimental Study of Molecular Mixing Enhancement in a Small Shear Layer Facility[1] , ROHIT NEHE, MANOOCHEHR KOOCHESFAHANI, Michigan State University — An experimental investigation is carried out to characterize the mixing field in a very low Reynolds number forced shear flow where the flow velocity is so low that, without the imposed perturbation, there is little mixing due to the absence of turbulence. To enhance the mixing interfacial area, we provide flow perturbation over a range of frequencies and amplitudes. The chemically reacting LIF technique is used to quantify the level of mixedness, while single component MTV is employed to measure the amplitude of perturbation velocity. Results from streamwise LIF measurements show the existence of a limited range of perturbation frequencies that result in a high level of mixedness. The highest mixing cases also exhibit high levels of velocity fluctuation. The 3D nature of the mixing field is studied in more detail by performing spanwise LIF measurements. Results will be presented for the cross-stream structure of the mixing field over the cross section of the test facility.

[1]Supported by the CRC Program of the National Science Foundation, Grant No. CHE-0714028.

3:46PM D11.00008 Numerical study of elastic turbulence in a 3D curvilinear micro-channel[1] , HONGNA ZHANG, TOMOAKI KUNUGI, Department of Nuclear Engineering, Kyoto University, FENGCHEN LI, School of Energy Science and Engineering, Harbin Institute of Technology — Elastic turbulence is an intriguing phenomenon of viscoelastic fluid flow, and dominated by the strong nonlinear elasticity due to the existence of flexible microstructures. It implies the possibility to generate a turbulent state (so-called an elastic turbulence) in the micro-scale devices by introducing the viscoelastic fluids, which could significantly enhance the mixing efficiency therein. Several experiments have been carried out to study its characteristics and underlying physics. However, the difficulty in measuring the flow information and behaviors of the microstructures, especially in the cross section normal to the mean flow direction, limits our current understanding and controlling. In the present study, the nondimensionalization method in which the characteristic velocity is defined as the ratio of the solution viscosity to the width of the channel was adopted to simulate the elastic turbulence in the micro-scale devices. And the elastic turbulent flow was obtained numerically in the 3D curvilinear micro-channel. Therein, the characteristics of the velocity field and polymer's behavior are discussed. Moreover, the energy transfer between the kinetic energy and the polymer's elastic energy is also investigated to understand its physical mechanism.

[1]Supported by the Japan Society for the Promotion of Science research fellowship and the Ministry of Education, Culture, Sports, Science and Technology via 'Energy Science in the Age of Global Warming' of Global Center of Excellence (G-COE) program (J-051)

3:59PM D11.00009 Electrothermal blinking vortices for chaotic mixing , SOPHIE LOIRE, PAUL KAUFFMANN, PAUL GIMENEZ, CARL MEINHART, IGOR MEZIC, UCSB — We present an experimental and theoretical study of electrothermal chaotic mixing using blinking of asymmetric 2D electrothermal vortices. Electrothermal flows are modelled with 2D finite element method using COMSOL software based on an enhanced electrothermal model. Velocities in top-view and side-view devices are measured by micro particle image velocimetry (μPIV). The experimentally reconstructed velocity profile shows a dramatic asymmetry between the two vortices, in good agreement with the FEM model. The separation line between the two vortices is shifted and tilted making the blinking vortices overlap. We use the mix-variance coefficient (MVC) on experimental particle detection data and 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. Theoretical, experimental and simulation results of the mixing process will be presented.

4:12PM D11.00010 Experimental Study of Electrothermal 3D Mixing using 3D microPIV , PAUL KAUFFMANN, SOPHIE LOIRE, CARL MEINHART, IGOR MEZIC, UCSB — Mixing is a keystep which can greatly accelerate bio-reactions. For thirty years, dynamical system theory has predicted that chaotic mixing must involve at least 3 dimensions (either time dependent 2D flows or 3D flows). So far, 3D embedded chaotic mixing has been scarcely studied at microscale. In that regard, electrokinetics has emerged as an efficient embedded actuation to drive microflows. Physiological mediums can be driven by electrothermal flows generated by the interaction of an electric field with conductivity and permittivity gradients induced by Joule heating We present original electrothermal time dependant 3D (3D+1) mixing in microwells. The key point of our chaotic mixer is to generate overlapping asymmetric vortices, which switch periodically. When the two vortex configurations blink, flows stretch and fold, thereby generating chaotic advection. Each flow configuration is characterized by an original 3D PIV (3 Components / 3 Dimensions) based on the decomposition of the flows by Proper Orthogonal Decomposition. Velocity field distribution are then compared to COMSOL simulation and discussed. Mixing efficiency of low diffusive particles is studied using the mix-variance coefficient and shows a dramatic increase of mixing efficiency compared to steady flow.

Sunday, November 18, 2012 2:15PM - 4:25PM –

Session D12 Vortex II 26B - Chair: Shilpa Khatri, University of North Carolina

2:15PM D12.00001 Vortex roll-up in a stratified fluid , SURUPA SHAW, JOHN MCHUGH, University of New Hampshire — Recent simulations of a vortex pair in a stratified fluid show that in some parameter regions the vortices disintegrate into internal waves. The kinetic energy loss for the vortex pair in this regime is remarkably fast, essentially annihilating the coherent vortex pair before any propagation. Hence the wave making occurs very early in the process. If the vortex pair is created by flow past a wing, then this wavemaking will occur mostly during the roll-up process of the trailing vorticity, and this is considered here. Results are obtained numerically 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 show that wavemaking is important over a much wider parameter range when the vorticity rolls-up in a stratified flow compared to previous simulation results. However for very strong vortex flows, there is no significant wavemaking, and the distributed vorticity very quickly rolls-up into a vortex pair.

2:28PM D12.00002 Experiments and numerical simulations of dense-core vortex rings in a sharply stratified environment[1] , RICHARD MCLAUGHLIN, ROBERTO CAMASSA, SHILPA KHATRI, KEITH MERTENS, CLAUDIO VIOTTI, University of North Carolina at Chapel Hill, Mathematics — Ambient stratification strongly influences the mixing and dispersion properties of particle dynamics. Much insight can be gained by studying the simplified setup of vortex ring dynamics. We present three dimensional direct numerical simulations of the dynamics for a vortex ring settling in sharply stratified miscible ambient fluids for two layer configurations. These simulations are compared with experiments conducted in the UNC Joint Fluids Lab. The core fluid of the vortex rings has density higher than both the top and bottom layers of the ambient fluid, and is fully miscible in both layers. This setup results in a rich parameter space which we partially present here. In particular, a critical (bifurcation) phenomenon is identified which distinguishes long-time behavior of the falling vortex ring. The ring either fully traps at the ambient density layer, or continues through the layer in its downward motion. This critical behavior is set by initial conditions (e.g., size and speed of the vortex ring, initial distance to the layer, etc.). Detailed comparisons between experiments and simulations for density, velocity, and vorticity fields will be discussed.

[1]This research is supported by NSF RTG DMS-0502266, NSF RTG DMS-0943851, NSF RAPID CBET-1045653, NSF CMG ARC-1025523, NSF DMS-1009750, and ONR DURIP N00014-09-1-0840.

2:41PM D12.00003 Structure of a vorticity patch bounded by a vortex sheet in strain[1] , DANIEL FREILICH, STEFAN LLEWELLYN SMITH, UCSD — Llewellyn Smith and Crowdy[*J. Fluid Mech.* **691** (2012)] studied the shape and stability of a constant pressure region bounded by a jump in the Bernoulli constnat, i. e. a vortex sheet, in an ambient irrotational straining field. We extend this work to the case of a two-dimensional vortex sheet bounding a uniform voricity patch, again in an ambient irrotational straining field. We obtain the relation between the two governing nondimensional parameters relating the strengths of the straining field, vortex sheet and vorticity in the patch, and examine the shape of the resulting vortex. We also investigate the lowest order correction for the shape of the vortex when the straining field is weak.

[1]Support from NSF-CMMI-0970113

2:54PM D12.00004 ABSTRACT WITHDRAWN –

3:07PM D12.00005 On the Anti-Symmetric Modes of Crow Instability by a General Instability Analysis Method , ZHONGQUAN CHARLIE ZHENG, University of Kansas, JAY HARDIN, NASA Langley Research Center (Retired) — The sinusoidal instability of a counter-rotation vortex pair has been investigated in Crow's seminal work (Crow, 1970). The anti-symmetric modes of instability were considered weak interaction modes by Crow, although they are the most amplified modes, according to Fig.11 in his paper. The weak interaction instability modes are those disturbances that are near the zero-self-induction and in the high wave-number range. However, later Saffman (1992) stated that all the anti-symmetric modes were stable. In this paper, the disturbance matrix is investigated. By looking at the eigenvalues and eigenvectors of the growth matrix, the symmetric and anti-symmetric modes of Crow's instability are recovered. Furthermore, by using a general instability analysis method of Farrell and Ioannou (1996), the upper bounds of the instability can be obtained, which again proves that the anti-symmetric modes are more amplified. These anti-symmetric modes can occur in both the long-wave modes and short-wave modes.

3:20PM D12.00006 Lifetime and layering of vortices in rotating stratified fluids , ORIANE AUBERT, MICHAËL LE BARS, PATRICE LE GAL, IRPHE - CNRS — Ocean and atmosphere are natural stratified fluid layers influenced by the rotation of the planet through the Coriolis force, where it is common to observe long-lived anticyclonic vortices sometimes surrounded by layers of constant density as the ocean Meddies. In the continuity of the experiments of Griffiths & Linden (1981) and Hedstrom & Armi (1988), we reproduce a rotating and linearly stratified layer in a tank where freely-decaying or sustained laboratory anticyclonic vortices are created via a short or continuous injection of isodense fluid. We quantify their long term evolution using PIV measurements. The Rossby number Ro of the freely-decaying vortices decreases in time, which is theoretically described by the energy conservation equations applied to a gaussian model that fits both laboratory and oceanic vortices. Using this theory and numerical simulations, we investigate the respective roles of rotation and stratification to explain the longevity of the vortices. Ro for the sustained vortices remains large and allows for the formation of layers above and below the vortices, following the double-diffusive instability of McIntyre (1970). Typical length and time scales of the instability are well described by a linear stability analysis based on our gaussian model.

3:33PM D12.00007 A bypass transition in the Lamb-Oseen vortex , LUIGI BISANTI, PIERRE BRANCHER, CHRISTOPHE AIRIAU, IMFT — Transient energy growth in the short-time linear dynamics of a Lamb-Oseen monopole is a potential mechanism for nonlinear bypass transition, a phenomenon already observed in both experiments and numerical simulations. In the present study, we investigate this scenario by means of a nonlinear optimal perturbation approach, i.e. by looking for the initial perturbation whose evolution satisfies the fully nonlinear Navier-Stokes equations and maximizes the energy gain at a given time horizon. Preliminary two-dimensional results show that, for small initial amplitudes, the optimal perturbation and growth mechanisms observed in the linear regime are recovered. More particularly, the time evolution of the $m = 2$ optimal perturbation leads to an elliptical core deformation of the monopole, which suggests a potential bypass scenario driven by the non-linear dynamics. This is confirmed by computations for larger initial perturbation amplitudes: the optimal perturbation is similar to that of the linear regime but a subcritical bifurcation to a quasi-steady, high-energy, rotating tripole is observed.

3:46PM D12.00008 Formation of type II vortex streets , ILDOO KIM, Agency for Defense Development, X.L. WU, University of Pittsburgh — In experiments in 2D soap film, we observe two kinds of vortex streets. The "type I" vortex street is stable and its Kármán ratio (the ratio of the transverse spacing to the longitudinal spacing of constituent vortices) is between 0.3 and 0.5. In contrast, the "type II" vortex street is characterized by its meta-stability and a much higher Kármán ratio, measured between 0.5 and 0.7. We studied the condition of formation of the type II vortex street by independently controlling two length scales of the system - the head-on width of the obstacle W and the thickness of the boundary layer δ before detachment. Our experiment suggests that the vortex street is type II when $\delta/W < 0.4$. The type II vortex street eventually evolves into the type I at downstream. The lifetime of this meta-stable configuration is strongly affected by the thickness δ of the boundary layer.

3:59PM D12.00009 From a Desingularized Vortex Sheet Model to a Turbulent Mixing Layer , UJJAYAN PAUL, RODDAM NARASIMHA, Jawaharlal Nehru Centre for Advanced Scientific Research (JNCASR) — The temporal mixing layer is studied using the model of a slightly perturbed vortex sheet which is unstable and tends to roll-up in a spiral. The flow is inviscid and incompressible. A point vortex model tends to evolve into a chaotic cloud of point vortices instead of a smooth double-branched spiral. The vortex sheet model is derived (in closed form) from the basic equations of vortex dynamics. The problem of finite time singularity is handled by a technique that invokes longitudinal circulation density diffusion along the sheet at singular points. The present model uses linear segments to interpolate the sheet. Although it is computationally involved compared to point vortices, the vortex sheet does not get distorted and rolls-up into a smooth double branched spiral. The accuracy of such simulations can be independently verified by using the laws of vortex dynamics and conserved quantities. We observe the growth of the two-dimensional shear layer with time and the merger of vortex like structures. The dependence of the mixing layer on the initial conditions is studied in detail and tries to answer the question whether the vortex sheet model yields a turbulent mixing layer.

4:12PM D12.00010 Vortex Dynamics in High Reynolds number, Acoustically Forced, Reacting Wakes near the Global Stability Boundary , BENJAMIN EMERSON, KELVIN MURPHY, TIM LIEUWEN, Georgia Institute of Technology — This abstract discusses results from a set of PIV experiments involving longitudinally acoustically forced, bluff body stabilized flames. It is well known that wakes stabilizing high density ratio flames are convectively unstable and thus are more receptive to acoustic forcing, while low density ratio wakes are globally unstable and thus tend to oscillate at a global mode frequency. When the wake is forced near its natural frequency, its peak response may shift to (or towards) the forcing frequency, a phenomenon known as lock-in. These experiments show that the longitudinal forcing launches a pair of symmetrically shed vortices from the bluff body, in contrast with the wakes natural asymmetry. As the lock-in phenomenon is approached, the symmetrically stimulated vortices convect downstream initially in their varicose configuration, but then stagger until they are arranged in a sinuous configuration. The axial position at which this staggering occurs is a strong function of how close the forcing frequency is to the natural frequency, and the amplitude of the forcing. This effect is made evident by the spatial distribution of vortical fluctuations, by ensemble averaged vorticity contours, and by the cross spectrum of the fluid dynamics on either side of the flow centerline. This staggering process, and the position at which it occurs, has important implications on the thermoacoustics of bluff body stabilized flames, as it governs the dynamics of the heat release and its receptivity to acoustic forcing.

Sunday, November 18, 2012 2:15PM - 4:25PM –

Session D13 Geophysical: General II 27A - Chair: Claudia Cenedese, Wood Hole Oceanographic Institution

2:15PM D13.00001 Nonlinear Scale Interactions and Energy Pathways in the Ocean , MATTHEW HECHT, HUSSEIN ALUIE, LANL, GEOFFREY VALLIS, KIRK BRYAN, Princeton/GFDL, MATHEW MALTRUD, ROBERT ECKE, BETH WINGATE, LANL — 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, barotropization, and Rossby wave generation.

2:28PM D13.00002 How Do You Determine Whether The Earth Is Warming Up?[1] , JUAN RESTREPO, DARIN COMEAU, HERMANN FLASCHKA, University of Arizona — How does one determine whether the high summer temperatures in Moscow of a few years ago was an extreme climatic fluctuation or the result of a systematic global warming trend? How does one perform an analysis of the causes of this summer's high temperatures in the US, if climate variability is poorly constrained? It is only under exceptional circumstances that one can determine whether a climate signal belongs to a particular statistical distribution. In fact, climate signals are rarely "statistical;" there is usually no way to obtain enough field data to produce a trend or tendency, based upon data alone. There are other challenges to obtaining a trend: inherent multi-scale manifestations, and nonlinearities and our incomplete knowledge of climate variability. We propose a trend or tendency methodology that does not make use of a parametric or a statistical assumption and it is capable of dealing with multi-scale time series. The most important feature of this trend strategy is that it is defined in very precise mathematical terms.

[1]With support from the NSF and the Gulf of Mexico Research Initiative.

2:41PM D13.00003 Mechanical energy budget for horizontal convection , BISHAKHDATTA GAYEN, ROSS W. GRIFFITHS, GRAHAM O. HUGHES, JUAN A. SAENZ, Australian National University — A three dimensional direct numerical simulation is performed to study horizontal convection in a long channel at a large Rayleigh number (of $O(10^{12})$). A different temperature is applied over each half of the channel base and the flow is allowed to reach a state of thermal equilibrium in which there is no net heat input. The circulation and temperature field accords with that observed in previous experiments and numerical simulations, and we focus on understanding horizontal convection from an energetics viewpoint. All terms in the mechanical energy budget can be evaluated explicitly, and we use this methodology to show how a strong circulation can be maintained despite the tight constraints on viscous dissipation in the flow established by previous work. An important conclusion is that horizontal convection represents a highly efficient mechanism of mixing in a stratified fluid.

2:54PM D13.00004 The Hydrodynamics of Iceberg Capsize Near a Glacier Terminus , J.C. BURTON, L.M. CATHLES, D.R. MACAYEAL, W.W. ZHANG, University of Chicago, J.M. AMUNDSON, University of Alaska Southeast, S. CORREA-LEGISOS, Centro de Estudios Científicos — Marine-terminating glaciers lose most of their mass into the ocean by calving icebergs. The largest icebergs are frequently observed to capsize as they calve, releasing enormous amounts of gravitational potential energy. During this process they may collide with the glaciers' terminius, producing teleseismic "glacial earthquakes" which can be detected by the Global Seismic Network. We use a combination of laboratory wave-tank experiments and numerical modeling to show that the contact and pressure forces exerted on the glacier terminus are strongly influenced by the hydrodynamics of the capsize process. In particular, we find that hydrodynamics can significantly increase the magnitude and duration of the contact force with the terminus, and that the earthquake magnitude, expressed as a twice-integrated force history, is not simply proportional to iceberg size. Our results highlight the difficulty of interpreting seismograms due to iceberg collisions.

3:07PM D13.00005 Laboratory experiments investigating the influence of fjord circulation 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, representative of Sermilik Fjord where Helheim Glacier terminates. Two fjord circulations are compared to a control experiment with no forced flow. The estuarine circulation is generated by introducing fresh water at melting temperatures from a source at the water free surface near the ice block representing the glacier. The wind driven circulation is generated by vertically displacing a solid block at the end of the tank opposite the ice block, mimicking the observed fjord circulation driven by wind events. The magnitude of both circulations can be systematically varied. The circulation pattern observed in the control and estuarine experiments is similar to those observed in previous studies. A thin boundary layer of cold melt water mixes with ambient waters and rises until it finds either the interface between the two layers, if in the bottom layer, or the free surface, if in the top layer. The results suggest that the melt water mainly deposits within the interior of the water column and not entirely at the free surface, as confirmed by field observations. In the wind driven experiments, the submarine melting of the glacier is enhanced and it increases with increasing wind frequency, suggesting that this circulation is more efficient in transporting heat to the glacier.

[1]Support was given by the National Science Foundation project OCE-113008 and the WHOI-Arctic Research Initiative.

3:20PM D13.00006 3D Baroclinic Vortices in Rotating Stratified Shear: from an Orange Great Red Spot to Planet Formation , PEDRAM HASSANZADEH, PHILIP MARCUS, UC Berkeley — The presence of horizontal shear strongly influences the dynamics of vortices in rotating stratified flows. Examples of such vortices are the Jovian vortices in zonal shear, and the vortices of the protoplanetary disks in strong Keplerian shear. Studying the physics of these vortices and their interaction with the environment requires high-resolution 3D simulations: ignoring the vertical direction or lack of enough resolution eliminates or changes important physical processes such as the secondary circulation. This ageostrophic flow might be essential in dust accretion and planet formation in protoplanetary disks, and a key in longevity, color, and color-change of Jovian vortices. For example, the very recent (July 2012) color change of the Great Red Spot to pale orange is most likely created by such secondary circulation. We have used high-resolution 3D numerical simulations of the Boussinesq equations to study 3D baroclinic vortices embedded in rotating stratified shear. We discuss the physics of their secondary circulation its ability to transport red chromphores and dust particles. We also present preliminary results on the interaction of vortices with shear, and show how this interaction affects their longevity.

3:33PM D13.00007 Zonal turbulence driven by baroclinic instabilities in the thermal quasi-geostrophic equations[1] , EMMA WARNEFORD, PAUL DELLAR, OCIAM, Mathematical Institute, University of Oxford — We present a mass- and momentum-conserving single layer model for the atmospheres of gas giant planets that produces both sub- and super-rotating equatorial jets. The thermal shallow water equations support horizontal temperature variations by treating the reduced gravity as an advected scalar. They were originally proposed for terrestrial lakes and tropical oceans. We add a radiative coupling term to create a model for the atmospheres of gas giant planets. Reducing the radiative relaxation time produces transitions from sub- to super-rotating equatorial jets. The radiative coupling also enhances the rate at which eddy kinetic energy supplied by small-scale random forcing is absorbed into the mean zonal flow. The quasigeostrophic limit of our model supports an instability whose dispersion relation coincides, up to an O(1) numerical factor, with the dispersion relation for baroclinic instability in continuously stratified fluids. This instability can drive sustained turbulence from imposed large-scale variations in the background temperature from pole to equator. Running our spectral numerical simulations on graphical processing units (GPUs) yields substantial (factors of ten) performance increases with little additional programming effort.

[1]Partially supported by EPSRC grant EP/E054625/1

3:46PM D13.00008 Zonal winds and flows generated by harmonic forcing in planetary atmospheres, subsurface oceans and cores , ALBAN SAURET, DAVID CEBRON, MICHAEL LE BARS, STEPHANE LE DIZES, PATRICE LE GAL, IRPHE, CNRS and Aix-Marseille University — A huge amount of energy is stored in the spin and orbital motions of any planet, and under certain circumstances, harmonic forcings such as libration, precession and tides are capable of conveying a portion of this energy to drive intense three-dimensional flows in its liquid layers. These mechanisms are studied here by combining theoretical, experimental and numerical approaches. At first, we focus on the effect of longitudinal librations, corresponding to oscillations of the rotation rate of a planet. This boundary forcing systematically leads to a correction to the mean solid body rotation through non-linear interactions in the Ekman layers. This geostrophic zonal wind is well described by an analytical approach. Additionally, at sufficiently large libration amplitude or small Ekman number, the oscillating flow is periodically unstable with respect to centrifugal instability. The resulting Taylor-Görtler vortices generated in the Ekman layers then generate inertial waves in the bulk with well-defined characteristics and temporal signatures. Inertial waves can also be resonantly excited by any harmonic forcing, when the forcing frequency ranges between plus and minus twice the rotation rate. In any case, the nonlinear self-interaction of excited inertial waves may drive an intense and localised axisymmetric jet, which becomes unstable at low Ekman number following a shear instability, generating space-filling turbulence. This generic mechanism is illustrated here by an experimental study of tidal forcing in a spherical shell.

3:59PM D13.00009 Coherent structures and warm-core rings in the Gulf of Mexico , DOUG LIPINSKI, KAMRAN MOHSENI, University of Florida — We use Lagrangian coherent structures (LCS) to investigate the three-dimensional structure of warm core rings shed from the loop current in the Gulf of Mexico. Using LCS allows for a precise computation of the eddies' depth that closely matches a model to predict eddy depth based on the geostrophic balance. Additionally, the LCS reveal a checkerboard pattern and interesting flow dynamics in the near surface boundary layer that causes fluid to be stretched and wrapped around the eddy. The flow behavior in this region is analyzed and compared to an analytically defined flow model where many properties may be proved directly. Notably, the relative strength of hyperbolic stretching and shear influences the transport and mixing properties of the flow.

4:12PM D13.00010 An experimental and numerical study of cyclones produced by suction in rotating stratified flows , PATRICE LE GAL, IRPHE - CNRS, PEDRAM HASSANZADEH, Berkeley University, ORIANE AUBERT, MICHAEL LE BARS, IRPHE - CNRS, PHILIP MARCUS, Berkeley University, IRPHE/BERKELEY COLLABORATION — Rotating and stratified flow motions are well-described by the gradient-wind equation, from which Hassanzadeh *et al.* (2012) and Aubert *et al.* (2012) derived a new law for the shape of the 3D vortices. The new equation was confirmed experimentally and numerically, and using the measurement data of the Atlantic Meddies and Jovian vortices. One consequence of this equation is that the interior of cyclones (anticyclones) must be more (less) stratified than the background flow. This means that to generate a cyclone (anticyclone) in nature, a process must produce both cyclonic (anticyclonic) vorticity and local-superstratification (a locally mixed patch of density). We have used laboratory experiments and 3D numerical simulations to study cyclones produced by localized suction, and we show that this process in fact produces both cyclonic vorticity and super-stratification. The physics of super-stratification and decaying cyclones in rotating stratified flows is studied. This brings a new understanding of the asymmetry between cyclones and anticyclones in nature, as it appears to be easier to locally mix some fluid to create a patch with a weaker stratification than the background as inside anticyclones, rather than to locally super-stratify it as inside cyclones.

Sunday, November 18, 2012 2:15PM - 4:25PM –

Session D14 General Experiments II 27B - Chair: Rinaldo Miorini, The Johns Hopkins University

2:15PM D14.00001 Novel Method to Characterize Superhydrophobic Coatings , GARY C. TEPPER, MOHAMED A. SAMAHA[1], HOOMAN VAHEDI TAFRESHI, MOHAMED GAD-EL-HAK, Virginia Commonwealth University — Superhydrophobic surfaces possess strong water-repellent characteristic, which, among several other potential applications, enhances the mobility of water droplets over the coatings. The surface entraps air within its micropores. When a coated body is submerged and in relative motion with water, shear-free and no-slip regions alternate over, respectively, the air pockets and the solid surface. The coating maintains its hydrophobicity as long as the air remains entrapped. It is therefore of great interest to precisely measure the amount of trapped air, which is particularly difficult to estimate for coatings with disordered microstructures. A novel method to measure the gas volume fraction of superhydrophobic coatings with either ordered or random microroughness is advanced. The technique is applied to both aerogel and electrospun-fibrous coatings. The experiments utilize a very sensitive weighing scale (down to 10^{-4} gm) and height gauge (down to 10 micron) to determine the buoyancy force on an immersed, coated glass-slide substrate. The measured force is used to calculate the volume fraction of entrapped air. *Effective* coating's thickness also follows from the same calculations.

[1]Presently at Princeton University

2:28PM D14.00002 Measuring Cavitation with Synchrotron X-Rays[1] , DANIEL DUKE, ALAN KASTENGREN, CHRIS POWELL, Argonne National Laboratory, X-RAY FUEL SPRAY GROUP, ENERGY SYSTEMS DIVISION TEAM — Cavitation plays an important role in the formation of sprays from small nozzles such as those found in fuel injection systems. A sharp-edged inlet from the sac into the nozzle of a diesel fuel injector is shown to inititate a strong sheet-like cavitation along the boundary layer of the nozzle throat, which is difficult to measure and can lead to acoustic damage. To investigate this phenomenon, a diagnostic technique capable of mapping the density field of the nozzle through regions of intense cavitation is required. Available visible-light techniques are limited to qualitative observations of the outer extent of cavitation zones. However, brilliant X-rays from a synchrotron source have negligible refraction and are capable of penetrating the full extent of cavitation zones. We present the early results of a novel application of line-of-sight, time-resolved X-ray radiography on a cavitating model nozzle. Experiments were conducted at Sector 7-BM of the Advanced Photon Source. Density and vapor distribution are measured from the quantitative absorption of monochromatic X-rays. The density field can then be tomographically reconstructed from the projections. The density is then validated against a range of compressible and incompressible numerical simulations.

[1]This research was performed at the 7-BM beamline of the Advanced Photon Source. We acknowledge the support of the U.S. Department of Energy under Contract No. DE-AC02-06CH11357 and the DOE Vehicle Technologies Program (DOE-EERE).

2:41PM D14.00003 ABSTRACT WITHDRAWN –

2:54PM D14.00004 3D effects in axisymmetric reconstructions , HANNA MAKARUK, CHRISTOPHER TOMKINS, Los Alamos National Lab — In a variety of penetrating diagnostics in fluid mechanics and elsewhere, quantities of interest are reconstructed from a single-view measurement under an assumption of axisymmetry. Here we employ theory, simulation and experiment to explore the effects of 3-dimensionality on these reconstructions. A key finding is that 3D effects may cause local negative densities, which are clearly unphysical, to appear in Abel inverse-type reconstructions. Analytical solutions are derived for violations of the axisymmetric assumption in the form of simple geometric shapes, and numerical Abel inversions are also performed on similar geometric problems. Both theory and numerics predict significant regions of inferred negative density under these conditions. These predictions are tested against an experimental measurement of known, idealized objects (spheres) using quantitative penetrating radiography. Results are compared for various values of the characteristic parameter, D/r (where D is the distance from reconstruction axis, r is the sphere radius). The results suggest that negative density values in Inverse-Abel reconstructions should not be ignored; instead, they provide potentially important insights into the 3D nature of the underlying phenomena.

3:07PM D14.00005 Performance of hot-wire probes designed to simultaneously measure three velocity components , RACHEL EBNER, CALEB MORRILL-WINTER, University of New Hampshire, RIO BAIDYA, University of Melbourne, PETAR VUKOSLAVCEVIC, University of Montenegro, JOSEPH KLEWICKI, University of New Hampshire and University of Melbourne, JAMES WALLACE, University of Maryland, NICHOLAS HUTCHINS, University of Melbourne — Vukoslavcevic (*Exp. in Fluids* **53**, 2012) recently used highly resolved channel flow DNS at low Reynolds number to investigate a number of hot-wire probe configurations for obtaining simultaneous measurements of all three velocity components. A focus of his effort was to minimize errors due to velocity gradients across the sensor array. A physical realization of Vukoslavcevic's XP (and XL) probe configuration was designed and fabricated. New fabrication techniques were implemented to minimize flow blockage and ensure uniform prong taper. In-situ pitch and yaw calibrations of the sensor are realized using a compact articulating jet that employs a novel flow-speed controller as the angle of the jet is varied. We present measurements derived from three facilities: the UNH 8m boundary layer wind tunnel, the University of Melbourne High Reynolds Number Boundary Layer Wind Tunnel (HRNBLWT), and the UNH Flow Physics Facility (FPF). The performance of the XL and XP probe configurations are assessed over a range of Reynolds numbers, as is the effect of pitching the probe at a fixed angle during calibration.

3:20PM D14.00006 Reconstructing dominant three-dimensional flow structures in the wakes of cylindrical bodies using planar velocity measurements[1] , CHRIS MORTON, SERHIY YARUSEVYCH, University of Waterloo — The flow over cylindrical bodies typically involves the periodic formation of spanwise vortices. Numerous experimental techniques have been employed in the past several decades for investigating the physical characteristics of wake vortices, e.g., surface pressure measurement with microphone arrays, velocity measurement with Laser Doppler Velocimetry (LDV) and Particle Image Velocimetry (PIV). More recently, direct numerical simulations and the advancements in volumetric measurement techniques have enabled quantitative investigation of the three-dimensional wake development. However, in the domain of physical experiments, planar velocity measurement systems still remain to be the mainstream tool. Thus, dominant three-dimensional flow features are commonly reconstructed by using phase-averaged planar measurements conducted at several planes, which requires a periodic reference signal related to flow development. The present investigation utilizes a novel approach for phase averaging 2D PIV data by extracting the reference signal via Proper Orthogonal Decomposition (POD) of the PIV data. The method is applied to investigate three-dimensional development of coherent structures in the wake of complex cylindrical geometries.

[1]The authors gratefully acknowledge the Natural Sciences and Engineering Research Council of Canada (NSERC) for funding of this work.

3:33PM D14.00007 In measurements of spatial distribution of surface deformation in studies of flow induced vibration[1] , CAO ZHANG, RINALDO MIORINI, JOSEPH KATZ, Johns Hopkins University — In studies involving flow-induced vibrations, it is necessary to measure the spatial distribution of surface motion simultaneously with the unsteady flow causing it. To achieve this goal, we developed a method for measuring the time resolved spatial distribution of surface deformation. Different approaches can be used for transparent and opaque surfaces. For a transparent wall, e.g. a compliant PDMS coating, high speed, digital Mach-Zehnder Interferometry maps the time-resolved surface shape. Calibrations demonstrate that a sub-micron resolution can be readily achieved, and that the sensitivity of the method can be adjusted to the expected range of surface deformations by varying the refractive index of the fluid. This technique is integrated with a time-resolved tomographic PIV system by bleeding/sampling 0.1% of the thick laser sheet energy illuminating the wall, and does not require an additional light source. Combined, it allows simultaneous flow and deformation measurements. For opaque surface, time resolved electronic speckle pattern interferometry (ESPI) could be used to measure the surface deformation by interfering light reflected from the surface with an external reference beam. Calibrations demonstrate that this method also achieves sub micron resolution.

[1]Sponsored by ONR

3:46PM D14.00008 Development of Luminescent Imaging for Capturing Cavitation in Water on Solid Surface , AKIHISA AIKAWA, JUN ANDO, Kyushu University, HIROTAKA SAKAUE, JAXA — Two-color pressure-sensitive paint (PSP) is applied to a solid surface to capture the cavitation acting on the surface in water. It is found that the luminescent signal increases under a cavitation region. The luminescence change of a PSP can be related to the oxygen quenching. Based on these, we discuss the luminescence increase at the cavitation region related to the oxygen concentration in water and oxygen pressure of a cavitation bubble. To extract the cavitation from an acquired luminescent image, the motion-capturing PSP method is applied. It eliminates the variation in illumination caused by the bubble creations between the PSP-coated surface and the imaging-acquisition instruments. The time-resolved cavitation images on the PSP-coated surface are captured inside an ultrasonicate bath.

3:59PM D14.00009 Global Temperature Measurement of Supercooled Water under Icing Conditions using Two-Color Luminescent Images and Multi-Band Filter , MIO TANAKA, Kanagawa Institute of Technology, KATSUAKI MORITA, The University of Tokyo, SHIGEO KIMURA, Kanagawa Institute of Technology, HIROTAKA SAKAUE, JAXA — Icing occurs by a collision of a supercooled-water droplet on a surface. It can be seen in any cold area. A great attention is paid in an aircraft icing. To understand the icing process on an aircraft, it is necessary to give the temperature information of the supercooled water. A conventional technique, such as a thermocouple, is not valid, because it becomes a collision surface that accumulates ice. We introduce a dual-luminescent imaging to capture a global temperature distribution of supercooled water under the icing conditions. It consists of two-color luminescent probes and a multi-band filter. One of the probes is sensitive to the temperature and the other is independent of the temperature. The latter is used to cancel the temperature-independent luminescence of a temperature-dependent image caused by an uneven illumination and a camera location. The multi-band filter only selects the luminescent peaks of the probes to enhance the temperature sensitivity of the imaging system. By applying the system, the time-resolved temperature information of a supercooled-water droplet is captured.

4:12PM D14.00010 Lifetime Characterization of Electro-Luminescence Based Pressure-Sensitive Paint System for Unsteady Flow Field Measurements , YOSHIMI IIJIMA, HIROTAKA SAKAUE, JAXA — Electro-luminescence based pressure-sensitive paint (EL-PSP) system uses an EL as an illumination source for a PSP measurement. EL can be directly applied onto a PSP model to eliminate a remote illumination. This gives a uniform illumination on a PSP model without moving/re-directing the illumination. The temperature dependency can be reduced by the opposite temperature dependency of the EL and PSP. At present, the system is demonstrated in a steady flow field. To extend the system for capturing an unsteady flow field, a fast responding PSP and the lifetime characterization of the system are required. The former can be achieved by using a porous PSP. The latter is discussed in the present presentation. The EL-PSP system needs an AC input to illuminate the EL, which gives a pulsed/periodic excitation to a PSP. This limits the acquisition timing of the flow field; a frequent timing can resolve a fast unsteady flow field. The lifetime of the PSP emission can be related to the pressure. The lifetime decays of the EL and PSP are measured to discuss the lifetime characterization of the system.

Sunday, November 18, 2012 2:15PM - 4:25PM –

Session D15 Biofluids: Large Swimmers I 28A - Chair: Ellen Longmire, University of Minnesota

2:15PM D15.00001 Dynamics of a self-propelled undulating swimmer[1] , SOPHIE RAMANANARIVO, MAXIME DANA, BENJAMIN THIRIA, RAMIRO GODOY-DIANA, PMMH UMR7636 CNRS, ESPCI ParisTech, UPMC (Paris 6), U. Paris Diderot (Paris 7) — Undulatory propulsion is a mean of locomotion shared by living organisms over a wide range of scales and in many different media. From eels to spermatozoa or motile bacteria, net forward motion is achieved by propagating backward, actively or passively, elastic waves along a deformable body. Here, we use a simple yet versatile experiment that constitutes a good framework to study the dynamics of undulatory swimmers. The set-up consists in a flexible filament forced to oscillate by imposing a harmonic motion to one of its extremities, and propelling itself at the free surface of a water tank. The present experiments pertain to the inertial regime for which Lighthill's elongated-body theory is the reference theoretical framework. We fully characterize the nature of the wave travelling the filament to understand the changes in the propulsive performance encountered in this inertial regime. We analyze in particular the role of the spatial envelope of the elastic wave, which is crucial in the present experiment where the oscillation of the filament is driven at the head of the swimmer and the deformation of the tail is passive.

[1]We acknowledge support from the French National Research Agency through project No. ANR-08-BLAN-0099 and of EADS Foundation through project "Fluids and elasticity in biomimetic propulsion."

2:28PM D15.00002 Fluid dynamics of forward swimming and turning for jellyfish , LAURA MILLER, University of North Carolina — Jellyfish propel themselves through the water through periodic contractions of their elastic bells. Some jellyfish, such as the box jellyfish Tripedalia cystophora and the upside down jellyfish Cassiopea xamachana, can perform turns via asymmetric contractions of the bell and by generating asymmetries in the outflow opening of the bell. The fluid dynamics of jellyfish forward propulsion and turning is explored here using the immersed boundary method. The 2D and 3D Navier-Stokes equations are coupled to the motion of a simplified jellyfish represented by an elastic boundary. An adaptive and parallelized version of the immersed boundary method (IBAMR) is used to resolve the detailed structure of the vortex wake. The asymmetric contraction and structure of the jellyfish generates asymmetries in the starting and stopping vortices. This creates a diagonal jet and a net torque acting on the jellyfish.

2:41PM D15.00003 Learning from real and tissue-engineered jellyfish: How to design and build a muscle-powered pump at intermediate Reynolds numbers , JANNA NAWROTH, Division of Biology, California Institute of Technology, HYUNGSUK LEE, ADAM FEINBERG, CRYSTAL RIPPLINGER, MEGAN MCCAIN, ANNA GROSBERG, School of Engineering and Applied Sciences, Harvard University, JOHN DABIRI, Graduate Aeronautical Laboratories and Bioengineering, California Institute of Technology, KIT PARKER, School of Engineering and Applied Sciences, Harvard University — Tissue-engineered devices promise to advance medical implants, aquatic robots and experimental platforms for tissue-fluid interactions. The design, fabrication and systematic improvement of tissue constructs, however, is challenging because of the complex interactions of living cell, synthetic materials and their fluid environments. In a proof of concept study we have tissue-engineered a construct that mimics the swimming of a juvenile jellyfish, a simple model system for muscle-powered pumps at intermediate Reynolds numbers with quantifiable fluid dynamics and morphological properties. Optimally designed constructs achieved jellyfish-like swimming and generated biomimetic propulsion and feeding currents. Focusing on the fluid interactions, we discuss failed and successful designs and the lessons learned in the process. The main challenges were (1) to derive a body shape and deformation suitable for effective fluid transport under physiological fluid conditions, (2) to understand the mechanical properties of muscle and bell matrix and device a design capable of the desired deformation, (3) to establish adequate 3D kinematics of power and recovery stroke, and (4) to evaluate the performance of the design.

2:54PM D15.00004 Viscous flow around a rapidly collapsing cylinder as a model of animal locomotion , GABRIEL WEYMOUTH, SMART, MICHAEL TRIANTAFYLLOU, MIT — A large body of experimental research indicates that shape change is instrumental in the locomotion of many animals from basilisk lizards to swifts and ducks. As a two dimensional model of such body shape changes, we examine the changes in force, energy, and vorticity induced by two manners of rapidly reduced cylinders; a "deflating" cylinder with prescribed kinematics, versus a prescribed "melting" cylinder similar to the problem of the vanishing disk considered by Taylor in 1953. Using large-scale viscous flow simulations, we show that the dynamics of the two cases generate fundamentally different flow fields. The deflating cylinder practically erases the memory of the original larger cylinder flow, with the excess kinetic energy being recovered at the body boundary, and opposite-sign vorticity cancels the excess boundary layer vorticity. In contrast, the melting cylinder case shows instantaneous and global shedding of the vorticity, which rapidly form into two strong vortices that contain the excess kinetic energy. Both the shrinking and melting body conditions are then used to demonstrate the effect of shape changing appendages in a set of simple two-dimensional maneuvering problems.

3:07PM D15.00005 ABSTRACT WITHDRAWN –

3:20PM D15.00006 A Three-Dimensional Multi-Domain Immersed Boundary Method, with Application to a Pitching Wing , CHENGJIE WANG, JEFF D. ELDREDGE, Mechanical and Aerospace Engineering, University of California, Los Angeles — A three-dimensional multi-domain technique and immersed boundary projection method is implemented for high-fidelity solution of the Navier-Stokes equations based on the approach presented by Colonius and Taira (2008). The principle of the multi-domain approach is to derive the boundary condition on a given domain from the interpolation of the solution on a larger, but coarser, mesh to simulate the unbounded flow. By performing this on a progression of such domains, the resulting flow in the original (finest) domain, which may contain some bodies, is able to account for the effect from the vorticity that is far away from it. On the other hand, the computational requirement is significantly relaxed compared to that of a single monolithic domain due to the compactness of each domain in the hierarchy, and the overall performance of the scheme is improved. The governing equations, and the immersed boundary treatment, are expressed in vorticity-streamfunction form. The resulting scheme is used to explore the physics of a low-aspect-ratio pitching wing in Re=100 flow. A wing of rectangular planform of aspect ratio 2 undergoes a steady pitch-up from 0 to 90 degrees in a uniform flow. Results for different pitching rate are compared and discussed.

3:33PM D15.00007 Modeling an elastic swimmer driven at resonance , PETER YEH, ALEXANDER ALEXEEV, George W. Woodruff School of Mechanical Engineering, Georgia Institute of Technology — Flexibility plays a vital role in the locomotion of aquatic animals. Using three dimensional computer simulations, we examine a flexible swimmer submerged in a viscous fluid with Reynolds number 100. The swimmer is modeled as a thin elastic rectangular plate, actuated at its leading edge to oscillate in a sinusoidal motion vertically at constant frequency and amplitude. The Lattice Boltzmann model is used to simulate an incompressible viscous fluid. The swimmer is free to move horizontally, and we measure the resulting steady state forward velocity, input power, and swimming performance. Our calculations reveal that both steady swimming velocity and performance strongly depend on the actuated frequency. Specifically, the maximum forward velocity is achieved near resonance, but the performance is maximized at a frequency about 1.8 times that at resonance. We visualize the vortex structures emerging in the fluid around swimmer and show how they contribute to the swimmer's forward motion.

3:46PM D15.00008 Applying IR Tomo PIV and 3D Organism Tracking to Study Turbulence Effects on Oceanic Predator-Prey Interactions[1] , DEEPAK ADHIKARI, MICHAEL HALLBERG, University of Minnesota, BRAD GEMMELL, Marine Biological Laboratory, ELLEN LONGMIRE, University of Minnesota, EDWARD BUSKEY, University of Texas at Austin — The behavorial response of aquatic predators and prey depends strongly on the surrounding fluid motion. We present a facility and non-intrusive instrumentation system designed to quantify the motions associated with interactions between small coral reef fish (blennies) and evasive zooplankton prey (copepod) subject to various flow disturbances. A recirculating water channel facility is driven by a paddlewheel to prevent damaging the zooplankton located throughout the channel. Fluid velocity vectors surrounding both species are determined by time-resolved infrared tomographic PIV, while a circular Hough transform and PTV technique is used to track the fish eye in three-dimensional space. Simultaneously, zooplankton motions are detected and tracked using two additional high-speed cameras with IR filters. For capturing larger scales, a measurement volume of 80 x 40 x 18 mm is used with spatial resolution of 3.5 mm. For capturing smaller scales, particularly for observing flow near the mouth of the fish during feeding, the measurement volume is reduced to 20 x 18 x 18 mm with spatial resolution of 1.5 mm. Results will be presented for both freshwater and seawater species.

[1]Supported by NSF IDBR grant #0852875.

3:59PM D15.00009 Quantitative analysis of the role of symmetry in biomimetic propulsive wakes[1] , VERONICA RASPA, RAMIRO GODOY-DIANA, BENJAMIN THIRIA, PMMH UMR 7636 CNRS-ESPCI-UPMC Paris 6, Paris 7 — We address here the understanding of how animal propulsion is related to flow physics in biomimetic locomotion. It is known that the wake pattern observed in a cross-section behind swimming or flying animals is typically characterized by the presence of periodical vortex shedding. However, depending on species, propulsive wakes are distinguished by their spatial ordering: symmetric (squid-like) or asymmetric (fish-like), with respect to the motion axis. We conducted a very precise experiment to analyse the role of the wake topology in propulsion generation. Self-propulsion is achieved by the flapping motion of two identical pitching rigid foils, separated by a distance d. By keeping the momentum input unchanged, we compared both symmetric and asymmetric flapping modes. For the entire parameters range, the symmetric squid-like mode proves to be more efficient for thrust generation than the fish-like asymmetrical one. We show that this difference is due to a pressure effect related to the ability of each wake to produce, or not, significant mixing in the near wake region.

[1]We acknowledge support from the French National Research Agency through project No. ANR-08-BLAN-0099 and of EADS Foundation through project "Fluids and elasticity in biomimetic propulsion."

4:12PM D15.00010 Effect of longitudinal ridges on the aerodynamic performance of a leatherback turtle model[1] , KYEONGTAE BANG, JOOHA KIM, HEESU KIM, SANG-IM LEE, HAECHEON CHOI, Seoul National University — Leatherback sea turtles (*Dermochelys coriacea*) are known as the fastest swimmer and the deepest diver in the open ocean among marine turtles. Unlike other marine turtles, leatherback sea turtles have five longitudinal ridges on their carapace. To investigate the effect of these longitudinal ridges on the aerodynamic performance of a leatherback turtle model, the experiment is conducted in a wind tunnel at $Re = 1.0 \times 10^5 - 1.4 \times 10^6$ (including that of real leatherback turtle in cruising condition) based on the model length. We measure the drag and lift forces on the leatherback turtle model with and without longitudinal ridges. The presence of longitudinal ridges increases both the lift and drag forces on the model, but increases the lift-to-drag ratio by 15 – 40%. We also measure the velocity field around the model with and without the ridges using particle image velocimetry. More details will be shown in the presentation.

[1]Supported by the NRF program (2011-0028032).

Sunday, November 18, 2012 2:15PM - 4:25PM –

Session D16 Biofluids: Arteries 28B - Chair: Michael Plesniak, The George Washington University

2:15PM D16.00001 Detection of multi-scale secondary flow structures using anisotropic 2D Ricker wavelets in a bent tube model for curved arteries[1] , DANIEL H. PLESNIAK, Haverford College, KARTIK V. BULUSU, MICHAEL W. PLESNIAK, The George Washington University — Interpretation of complex flow patterns observed in this study of a model curved artery required characterization of multiple, low-circulation secondary flow structures that were observed during the late systolic deceleration and diastolic phases under physiological inflow conditions. Phase-locked, planar vorticity PIV data were acquired at various cross-sectional locations of the 180-degree bent tube model. High circulation, deformed Dean- and Lyne-type vortices were observed during early stages of deceleration, while several smaller scale, highly deformed, low-circulation vortical patterns appeared in the core and near-wall regions during late systolic deceleration and diastolic phases. Due to the multiplicity of vortical scales and shapes, anisotropic 2D Ricker wavelets were used for coherent structure detection in a continuous wavelet transform algorithm (PIVlet 1.2). Our bio-inspired study is geared towards understanding whether optimizing the shape of the wavelet kernel will enable better resolution of several low-circulation, multi-scale secondary flow morphologies and whether new insights into the dynamics of arterial secondary flow structures can accordingly be gained.

[1]Supported by the National Science Foundation, Grant No. CBET-0828903 and GW Center for Biomimetics and Bioinspired Engineering (COBRE).

2:28PM D16.00002 Perturbation-induced secondary flow structures due to fractured stents in arterial curvatures[1] , KARTIK V. BULUSU, CHRISTOPHER POPMA, LEANNE PENNA, MICHAEL W. PLESNIAK, The George Washington University — An in vitro experimental investigation of secondary flow structures was performed downstream of a model stent that embodied a "Type-IV" stent fracture, i.e. complete transverse fracture of elements and element displacement (of 3 diameters). One part of the fractured stent was located in the curved region of a test section comprised of a 180-degree bent tube, and the velocity field measured with PIV. Secondary flow morphologies downstream of the stent were identified with a continuous wavelet transform (CWT) algorithm (PIVlet 1.2) using a 2D Ricker wavelet. A comparison of wavelet transformed vorticity fields of fractured and unfractured model stents is presented under physiological inflow conditions. During systolic deceleration, a breakdown in symmetry of vortical structures occurred with the unfractured stent, but not with the fractured model stent. Potential mechanisms to explain the differences in secondary flow morphologies include redirection of vorticity from the meridional plane of the bend to the normal plane and diffusion of vorticity.

[1]Supported by the National Science Foundation, Grant No. CBET-0828903 and GW Center for Biomimetics and Bioinspired Engineering (COBRE).

2:41PM D16.00003 A regime map for secondary flow structures under physiological and multi-harmonic inflow through a bent tube model for curved arteries[1] , SHANNON M. CALLAHAN, KIRIN CALDWELL, KARTIK V. BULUSU, MICHAEL W. PLESNIAK, The George Washington University — Secondary flow structures are known to affect wall shear stress, which is closely related to atherogenesis and drug particle deposition. A regime map provides a framework to examine phase-wise variations in secondary flow structures under physiological and multi-harmonic inflow waveforms under conditions of a fixed Womersley number (4.2) and curvature ratio (1/7). Experimental PIV data were acquired at the 90-degree location in a 180-degree curved test section of a bent tube model for curved arteries using a blood analog working fluid. Coherent structure detection was performed using a continuous wavelet transform algorithm (PIVlet 1.2) and further analysis was carried out by grouping similar secondary flow structures at a fixed secondary Reynolds numbers. Phase-locked, planar vorticity fields over one period of inflow waveform revealed size, structure and strength similarities in secondary flow morphologies during the acceleration and deceleration phases. The utility of the new regime map lies in the a priori identification of pulsatile secondary flow structures, eliminating the need for exhaustive experimentation or computing, requiring only flow rate measurements that are easily acquired under clinical conditions.

[1]Supported by the National Science Foundation, Grant No. CBET-0828903 and GW Center for Biomimetics and Bioinspired Engineering (COBRE).

2:54PM D16.00004 A viscoelastic model of shear-induced blood damage[1] , GILAD ARWATZ, ALEXANDER SMITS, Princeton University — The mechanisms responsible for blood damage (hemolysis) have been studied since the mid-1960s, and it is now widely accepted that the level of shear stress and exposure time play important roles. Several models for hemolysis have been previously proposed. However, these models are purely empirical and limited to a narrow range of shear stress and exposure time and mostly, they lack any physical basis. In this study, we propose a new non-dimensional model that captures the mechanics of the red blood cells breakdown by taking into account the viscoelastic nature of their membrane. We validate our model against experimental measurements of hemolysis caused by laminar shear stress ranging from 50Pa to 500 Pa and exposure times extending from 60 s to 300 s.

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

3:07PM D16.00005 Volumetric lattice Boltzmann simulation for blood flow in aorta arteries , DEBANJAN DEEP, HUIDAN (WHITNEY) YU, Mechanical Engineering Department, Indiana University-Purdue University Indianapolis (IUPUI), SHAWN TEAGUE, Department of Radiology and Imaging Sciences, School of Medicine, Indiana University — Complicated moving boundaries pose a major challenge in computational fluid dynamics for complex flows, especially in the biomechanics of both blood flow in the cardiovascular system and air flow in the respiratory system where the compliant nature of the vessels can have significant effects on the flow rate and wall shear stress. We develop a computation approach to treat arbitrarily moving boundaries using a volumetric representation of lattice Boltzmann method, which distributes fluid particles inside lattice cells. A volumetric bounce-back procedure is applied in the streaming step while momentum exchange between the fluid and moving solid boundary are accounted for in the collision sub-step. Additional boundary-induced migration is introduced to conserve fluid mass as the boundary moves across fluid cells. The volumetric LBM (VLBM) is used to simulate blood flow in both normal and dilated aorta arteries. We first compare flow structure and pressure distribution in steady state with results from Navier-Stokes based solver and good agreements are achieved. Then we focus on wall stress within the aorta for different heart pumping condition and present quantitative measurement of wall shear and normal stress.

3:20PM D16.00006 Patient specific flow dynamic simulations of flow in diseased coronary artery , CARLOS MORENO, KIRAN BHAGANAGAR, University of Texas, San Antonio — Patient specific simulations of patients belonging to type I: protruding, type II: ascending, type III: descending, and type IV: diffuse have been performed to understand the effect of inlet forcing frequency and amplitude on the wall shear stress (WSS). Numerical simulations are performed with unsteady flow conditions in a laminar regime. The results have revealed that at low amplitudes, the sensitivity of WSS to forcing frequency is strongly dependent on the patient type for same degree of stenosis. For all the types, the maximum WSS is observed in post-stenotic or the distal region of the stenosis, and WSS has lowest magnitude at the peak location of the stenosis. For higher pulsatile amplitude ($a > 1.0$), WSS exhibits a strong sensitivity with forcing frequencies for all types. However, at higher forcing frequency the WSS exhibits nonlinear response to the inlet forcing frequency, which is strongly type dependent. The study clearly demonstrated differences between the intra-type flows are small compared to the inter-type flows.

3:33PM D16.00007 Numerical Investigation of Heat Transfer and Flow Characteristics of non-Newtonian Blood Flow in Atherosclerosis Coronary Artery: the Effect of Magnetic Field , SIAVASH GHAFFARI, University of Tehran, SHIMA ALIZADEH, Stanford University, MOHAMMAD SADEQ KARIMI, University of Tehran — Temperature heterogeneity in plaque containing inflammatory cells can cause thermal stress, and accelerates rupture process. Activated inflammatory cells embedded in plaques release heat while the plaque is cooled by blood flow. In the present work, arterial wall temperature distribution of atherosclerotic Right Coronary in the presence of external uniform and multi-directional magnetic field is investigated by numerical methods. The rheology of the flowing blood is modeled by a generalized Power law model. An advanced coupled FEM-FVM algorithm is used to determine temperature distribution inside the artery. Transient Navier-Stokes and energy equations in 2D idealized arterial model of a bending artery coupled with Maxwell's equations are discretized using the Finite-Volume Method and solved by SIMPLE algorithm in curvilinear coordinate to analyze pulsatile blood flow, whereas the transient heat conduction equation in the plaque is solved simultaneously with these equations using Finite-Element Method. The plaque temperature, Nusselt Number and heat flux at the plaque/lumen interface is obtained for different states of magnetic field and different Power law indices (n) to investigate influence of produced electromagnetic force and blood viscosity on the cooling effect of blood. It is observed that how magnetic field and blood dilution modifies the temperature heterogeneity of plaque and decreases probability of rupture of Atherosclerotic plaque.

3:46PM D16.00008 Wall Shear Stress in Aorta with Coarctation and Post-Stenotic Dilatation - Scale Resolved Simulation of Pulsatile Blood Flow , ROLAND GARDHAGEN, MATTS KARLSSON, Department of Management and Engineering, Linkoping University — Large eddy simulations of pulsating blood flow in an idealized model of a human aorta with a coarctation and a post-stenotic dilatation were conducted before and after treatment of the stenosis using Ansys Fluent. The aim was to study wall shear stress (WSS), which influences the function of endothelial cells, and turbulence, which may play a role in thrombus formation. Phase average values of WSS before the treatment revealed high shear in the stenosis at peak systole, as expected, but also at the end of the dilatation. In the dilatation backflow causes a negative peak. Diastolic WSS is characterized by low amplitude oscillations, which promotes atherogenesis. Also noticeable is the asymmetric pattern between the inner and outer sides of the vessel caused by the arch upstream of the stenosis. Thus, large spatial, temporal, and probably asymmetric WSS gradients in the already diseased region suggest increased risk for further endothelial dysfunction. This reflects a complex, partly turbulent, flow pattern that may disturb the blood flow in the abdominal aorta. After treatment of the stenosis, but not the dilatation, fluctuations of velocity and WSS were still found, thus harmful flow conditions still exist.

3:59PM D16.00009 Blood flow characteristics in the aortic arch[1] , LISA PRAHL WITTBERG, STEVIN VAN WYK, MIHAI MIHAIESCU, LASZLO FUCHS, Linne FLOW center, KTH Mechanics, Stockholm, Sweden, EPHRAIM GUTMARK, Dept. of Aerospace Engineering, University of Cincinnati, OH, USA, PHILIPPE BACKELJAUW, IRIS GUTMARK-LITTLE, Dept. of Pediatrics, Cincinnati Childrens Hospital, OH, USA — The purpose with this study is to investigate the flow characteristics of blood in the aortic arch. Cardiovascular diseases are associated with specific locations in the arterial tree. Considering atherogenesis, it is claimed that the Wall Shear Stress (WSS) along with its temporal and spatial gradients play an important role in the development of the disease. The WSS is determined by the local flow characteristics, that in turn depends on the geometry as well as the rheological properties of blood. In this numerical work, the time dependent fluid flow during the entire cardiac cycle is fully resolved. The Quemada model is applied to account for the non-Newtonian properties of blood, an empirical model valid for different Red Blood Cell loading. Data obtained through Cardiac Magnetic Resonance Imaging have been used in order to reconstruct geometries of the the aortic arch. Here, three different geometries are studied out of which two display malformations that can be found in patients having the genetic disorder Turner's syndrome. The simulations show a highly complex flow with regions of secondary flow that is enhanced for the diseased aortas.

[1]The financial support from the Swedish Research Council (VR) and the Sweden-America Foundation is gratefully acknowledged.

4:12PM D16.00010 Fluid-structure interaction analysis on the effect of vessel wall hypertrophy and stiffness on the blood flow in carotid artery bifurcation[1] , SANG HOON LEE, Seoul National University, HYOUNG GWON CHOI, Seoul National University of Science and Technology, JUNG YUL YOO, Seoul National University — The effect of artery wall hypertrophy and stiffness on the flow field is investigated using three-dimensional finite element method for simulating the blood flow. To avoid the complexity due to the necessity of additional mechanical constraints, we use the combined formulation which includes both the fluid and structural equations of motion into single coupled variational equation. A P2P1 Galerkin finite element method is used to solve the Navier-Stokes equations for fluid flow and arbitrary Lagrangian-Eulerian formulation is used to achieve mesh movement. The Newmark method is employed for solving the dynamic equilibrium equations for linear elastic solid mechanics. The pulsatile, incompressible flows of Newtonian fluids constrained in the flexible wall are analyzed with Womersley velocity profile at the inlet and constant pressure at the outlet. The study shows that the stiffness of carotid artery wall affects significantly the flow phenomena during the pulse cycle. Similarly, it is found that the flow field is also strongly influenced by wall hypertrophy.

[1]This work was supported by Mid-career Researcher Program and Priority Research Centers Program through the National Research Foundation of Korea (NRF) funded by the Ministry of Education, Science and Technology (2009-0079936 & 2011-0029613).

Sunday, November 18, 2012 2:15PM - 4:25PM –

Session D17 Biofluids: Microswimmers and Hydrodynamic Interactions 28C - Chair: Raymond Goldstein, University of Cambridge

2:15PM D17.00001 *E. coli* in a wall bounded shear flow[1] , MEHDI MOLAEI, JIAN SHENG, Texas Tech University — Understanding bacteria motility over a wall in a shear flow is critical to determine those crucial biophysical processes involved in the biofilm formation and the shear erosion. Using digital holographic microscopy combined with microfluidics we capture three-dimensional swimming patterns of wild-type *E. coli* bacteria in a straight micro-channel subjecting to a carefully controlled flow shear. Three-dimensional locations and orientations of bacterial are extracted with a resolution of 0.185 μm in lateral directions and 0.5 μm in the wall normal direction. Robust statistics based on thousands of trajectories allow us to characterize bacteria swimming over a surface under flow shear. These characteristics, including swimming velocity, tumbling frequencies, cellular attachment, and suspension dispersion, will be used to elucidate the cell wall interactions in shear flows. Current analysis will focus on the hydrodynamic mechanisms other than near field interfacial forces on cell migration and orientation near a sheared surface. Preliminary data on bacteria over a chemically modified surface will also be presented.

[1]National Institution of Health

2:28PM D17.00002 Bacteria motility at oil-water interfaces , GABRIEL JUAREZ, STEVEN SMIRGA, VICENTE FERNANDEZ, ROMAN STOCKER, Massachusetts Institute of Technology — The swimming dynamics of bacteria are strongly influenced by interfaces: Motile bacteria often accumulate at rigid boundaries, such as liquid-solid interfaces, and at soft boundaries, such as liquid-air or liquid-liquid interfaces. Attachment of bacteria to these interfaces is crucial for the formation of biofilms (liquid-solid), pellicles (liquid-air), and oil-degrading communities (liquid-liquid). We investigated the motility of the oil-degrading bacteria *Marinobacter aquaeolei* in the presence of oil droplets. We created individual oil droplets using dedicated microfluidic devices and captured the swimming behavior of individual bacteria near the interface and their attachment dynamics to the droplets with high-speed and epifluorescent microscopy. We find that *Marinobacter aquaeolei* has a high affinity towards interfaces and their swimming dynamics at soft interfaces differ from both those in the bulk and at rigid boundaries. Characterizing the interaction and attachment of motile bacteria to liquid-liquid interfaces will promote a fundamental understanding to oil-microbe interactions in aquatic environments and potentially lead to improved oil bioremediation strategies.

2:41PM D17.00003 Accumulation of swimming bacteria near an interface[1] , JAY TANG, GUANGLAI LI, Brown University — Microbes inhabit planet earth over billions of years and have adapted to diverse physical environment of water, soil, and particularly at or near interfaces. We focused our attention on the locomotion of Caulobacter crescentus, a singly flagellated bacterium, at the interface of water/solid or water/air. We measured the distribution of a forward swimming strain of C. crescentus near a surface using a three-dimensional tracking technique based on dark field microscopy and found that the swimming bacteria accumulate heavily within a micrometer from the surface. We attribute this accumulation to frequent collisions of the swimming cells with the surface, causing them to align parallel to the surface as they continually move forward. The extent of accumulation at the steady state is accounted for by balancing alignment caused by these collisions with rotational Brownian motion of the micrometer-sized bacteria. We performed a simulation based on this model, which reproduced the measured results. Additional simulations demonstrate the dependence of accumulation on swimming speed and cell size, showing that longer and faster cells accumulate more near a surface than shorter and slower ones do. The overarching goal of our study is to describe interfacial microbial behavior through detailed analysis of their motion.

[1]We acknowledge support by NSF PHY 1058375.

2:54PM D17.00004 Bacterial trapping in shear , ROBERTO RUSCONI, JEFFREY S. GUASTO, ROMAN STOCKER, MIT — Bacteria are ubiquitously exposed to flow, both in natural environments and artificial devices (e.g., catheters), where confining surfaces create non-uniform shear. While the effects of shear on passive particles are well understood, little is known about the consequences of shear on motile bacteria. We exposed bacteria having different motility strategies (e.g., run-and-tumble, run-and-reverse) to microfluidic Poiseuille flows and quantified the swimming kinematics and cell distribution in the channel using video-microscopy. We discovered that the coupling of motility and a spatially varying shear results in a dramatic trapping of motile cells in high-shear regions, and conversely a strong depletion in the low-shear portion of the channel. We demonstrate experimentally that this trapping process is robust across species such as *Bacillus subtilis* and *Pseudomonas aeruginosa*, and can have far-reaching consequences on bacterial transport, by (i) counteracting bacterial chemotactic responses; and (ii) enhancing surface attachment and thus biofilm formation by trapping cells near walls. More generally, this work shows that–despite the low Reynolds number–the coupling of flow and self-propulsion can be nonlinear and not simply a superposition of the two effects.

3:07PM D17.00005 Swimming of *E. coli* near micro-structured surfaces , VASILY KANTSLER, JORN DUNKEL, RAYMOND E. GOLDSTEIN, DAMTP, University of Cambridge — Understanding the mechanisms that govern surface accumulation of swimming bacteria is a key challenge for controlling biofilm formation. Here, we report detailed measurements of density and orientation distributions for *Escherichia coli* bacteria as a function of the distance from a solid surface. Experiments were performed for wild-type and non-tumbling strains in both quasi-2D and 3D microfluidic chambers. We find that, for both geometries, the density profile in dilute suspensions decays sharply within a few microns from flat surfaces approaching a constant value in the bulk. Our measurements of the orientation distributions show that bacteria preserve memory of aligning collisions with surfaces for surprisingly long periods of time even after escaping into the bulk fluid. These experimental results agree well with numerical simulations of a minimal mechanistic model that accounts for steric interactions between bacteria and surfaces. We further demonstrate that optimal micro-scale surface patterning can substantially decrease accumulation of swimming bacteria, thereby providing a novel mechanism for preventing biofilm formation.

3:20PM D17.00006 Helical swimming in confined geometries , KENNETH S. BREUER, BIN LIU, THOMAS R. POWERS, Brown University — We discuss how bacterial swimming is affected by spatial confinement at the micron scale, as in a porous medium. We model a bacterial swimmer in a porous medium by a rotating rigid helix in a cylindrical cavity with smooth walls. A novel boundary element method is introduced to make full use of the helical symmetry. This method allows us to investigate situations of tight confinement in which the helix comes very close to the walls. We show that the confinement enhances the swimming efficiency, especially when the circumference of the tube matches the contour length of one helical pitch. To our surprise, at fixed power consumption, a highly-coiled swimmer swims faster in a narrower confinement, while a more open coil swims faster in a cavity with a wider opening.

3:33PM D17.00007 Hydrodynamic synchronization of flagella on the surface of the colonial alga *Volvox carteri* , DOUGLAS BRUMLEY, MARCO POLIN, RAYMOND GOLDSTEIN, TIMOTHY PEDLEY, DAMTP, University of Cambridge — Whether on the surface of unicellular ciliates or in the respiratory epithelium, groups of eukaryotic cilia and flagella are capable of coordinating their beating over large scales. The mechanism responsible for the emergence of these metachronal waves is still unclear, mostly because finding an experimental system in which the beating filaments can be followed individually is challenging. We propose the multicellular green alga *Volvox carteri* as an ideal model system to study metachronal coordination, and report the existence of robust metachronal waves on its surface. Inspired by flagellar tip trajectories of *Volvox* somatic cells, we model a flagellum using a sphere of radius a elastically bound to a circular orbit of radius r_0, perpendicular to a no-slip plane. This elastohydrodynamic model of weakly-coupled self-sustained oscillators can be recast in terms of interacting phase oscillators, offering an intuitive understanding of the mechanism driving the emergence of coordination. Our results confirm that elasticity is fundamental to guarantee fast and robust synchronization, and that sufficiently compliant trajectories lead to the emergence of metachronal waves in a manner essentially independent of boundary conditions.

3:46PM D17.00008 Do proximate, C. elegans swimmers synchronize their gait?[1] , JINZHOU YUAN, DAVID RAIZEN, HAIM BAU, University of Pennsylvania — We imaged two C. elegans swimming, one after the other, in a tapered conduit. The conduit was subjected to a DC electric field, with the negative pole at the narrow end and applied flow directed from the narrow end. As a result of their attraction to the negative pole (electrotaxis), both animals swam upstream. As the conduit narrowed, the average adverse flow velocity increased and the swimming speed of the leading animal decreased faster than that of the trailing animal, allowing the latter to catch up with the former. We quantified synchronization by measuring the phase lag between the gait of one animal and the extended wave pattern of the other as a function of the distance between the two animals. Only when the distance between the two animals' body centers was nearly equal to or smaller than one body length were the animals' motions synchronized. When the nematodes were parallel to one another, synchronization was essential to prevent the animals from colliding. Direct numerical simulations indicate that when the trailing animal's head is immediately downstream of the leading animal's tail, the animals derive just a slight hydrodynamic advantage from their proximity compared to a single swimmer.

[1]We thank Kun He Lee from the University of Pennsylvania for preparing C. elegans.

3:59PM D17.00009 Simulations of artificial swimmers in confined flows[1] , LUCA BRANDT, LAILAI ZHU, EERIK GJØLBERG, KTH Mechanics, Royal Institute of technology — Miniature swimmming robots are potentially powerful for microobject manipulation, such as flow control in lab-on-a-chip, localized drug delivery and screening for diseases. Magnetically driven artificial bacterial flagella (ABF) performing helical motion is advantegous due to high swimming speed and accurate control. Using boundary element method, we numerically investigate the propulsion of ABF in free space and near solid boundaries. Step-out at high actuation frequencies, wobbling and near-wall drifting are documented, in qualitative agreement with recent experiments. We aim to explore the effect of swimmer shape on the performance, thus benefiting design of efficient microswimmers. Propulsion of ABF confined by a solid wall with and without background shear flow is also studied, with a focus on wall-induced hydrodynamic interaction and its influence on the stability of the motion.

[1]Funding by VR (the Swedish Research Council) and Linne flow centre at KTH is acknowledged.

4:12PM D17.00010 Hydrodynamical entrapment of ciliates at the air-liquid interface , JONATHAN FERRACCI[1], HIRONORI UENO, KEIKO NUMAYAMA-TSURUTA, Department of Biomedical Engineering, Tohoku University, YOHSUKE IMAI, Department of Bioengineering and Robotics, Tohoku University, TAKAMI YAMAGUCHI, Department of Biomedical Engineering, Tohoku University, TAKUJI ISHIKAWA, Department of Bioengineering and Robotics, Tohoku University — We found the new phenomenon of the entrapment of ciliates at the air-water interface, though they are not trapped by a solid interface. We first characterize the behaviours of cells at the interface by comparing it to those away from interfaces. The results showed that the cell's swimming velocity is considerably reduced at the air-water interface. In order to experimentally verify the possible physiological causes of the entrapment, we observed their behaviours in absence of positive chemotaxis for oxygen and the negative geotaxis. The results illustrated that the entrapment phenomenon was not dependent on these physiological conditions. The experiments using surfactant revealed that the entrapment phenomenon was strongly affected by the velocity-stress conditions at the interface. This fact was confirmed numerically by a boundary element method, i.e. the stress-free condition at the air-liquid interface is one of the main mechanisms of the entrapment phenomenon found in the experiments. Since the entrapment phenomenon found in this study affects the cell-cell interactions and the mass transport at the interface, the knowledge obtained in this study is useful for better understanding the complex behaviours of swimming microorganisms in nature.

[1]PhD student in the Physiological Flow Studies Laboratory.

Sunday, November 18, 2012 2:15PM - 4:25PM –

Session D18 Biofluids: Complex Interaction 28D - Chair: Roland Bouffanais, Singapore University of Technology and Design

2:15PM D18.00001 The effect of extracellular conductivity on electroporation-mediated molecular delivery[1] , MIAO YU, MOHAMED SADIK, JIANBO LI, HAO LIN, Mechanical and Aerospace Engineering, Rutgers University — Electroporation is a non-viral technique to introduce foreign molecules into a biological cell with electric fields. In this work, the effect of extracellular conductivity on electroporation-mediated molecular delivery efficiency is numerically investigated, and the results are compared in details with experimental data. The model couples the Smoluchowski equation and the Nernst-Planck equations to solve for the evolution of membrane permeabilization and ion transport simultaneously. The uptake of Propidium Iodide (PI) into single 3T3 fibroblast cells is simulated. The results quantitatively predict the experimental observations, and that the total delivery is reciprocally correlated with the extracellular conductivity. This correlation is mediated primarily by electrophoretic transport induced by a gradient in the electric field. A compact formula is also developed to estimate ion delivery based on the pulsing parameters. This work offers a mechanistic interpretation to experimental observations. In synergy with experimental efforts, the quantification of molecular delivery via electroporation has been achieved.

[1]NSF CBET 0747886 and CBET 0967598

2:28PM D18.00002 The compliance of vascular endothelial cells (VECs) change after exposure to cyclic, uniaxial stretch , KATHRYN OSTERDAY, Mechanical and Aerospace Engineering; University of California, San Diego (UCSD), THOMAS CHEW, LOURY PHILLIP, Bioengineering; UCSD, JASON HAGA, Bioengineering and Institute For Engineering in Medicine; UCSD, MANUEL GOMEZ-GONZALEZ, Mechanical and Aerospace Engineering; UCSD, JUAN CARLOS DEL ALAMO, Mechanical and Aerospace Engineering and Institute For Engineering in Medicine; UCSD, SHU CHIEN, Bioengineering, Institute For Engineering in Medicine and School of Medicine; UCSD — In vivo, VECs are exposed to both shear stress and cyclic, uniaxial stretch. It is known that VECs remodel their cytoskeleton perpendicular to stretch and parallel to shear and that cytoskeletal structure is critical to vessel function. Cytoskeletal structure must affect the magnitude and direction of the maximum and minimum shear compliance of the cytoplasm. This may provide the cell with a mechanism to tune their sensitivity to external mechanical stimuli differently along different directions, providing the flow-sensing mechanism needed for mechanotransduction. To study how cytoskeletal remodeling is correlated to changes in subcellular microrheology, we used directional particle tracking microrheology (DPTM) to calculate the shear compliance of the cytoplasm before and after exposure to cyclic, uniaxial stretch. When stretched, we find, VECs align their direction of maximum shear compliance perpendicular to stretch, their cytoplasm becomes less liquid, and the magnitude of the shear compliance along both directions of mechanical polarization decrease.

2:41PM D18.00003 Fluid coupling in continuum modeling of microtubule motility assays , CHRISTEL HOHENEGGER, University of Utah, TAMAR SHINAR, University of California, Riverside — Active networks are suspensions of actuated filaments obtained by mixing cytoskeletal filaments and small motor protein complexes. The network activity is achieved by the motion of the motor proteins along the filaments. Here, we focus on motility assays, where the molecular motors are anchored to a bottom plate and filaments are observed to glide in a quasi two-dimensional plane. In contrast to other studies, we are not interested in the detailed motion of each individual filament but only in the resulting coarse grained properties. We present a new continuum macroscopic model of motility assays including the evolution of rigid filaments density, bound and free motors densities and fluid velocity. The coupling between the fluid and the filaments is described via external, localized molecular forces inspired from the immersed boundary method. The reduction to a two-dimensional system in the plane of motion of the filaments is achieved via depth-averaging similar to Hele-Shaw approximation. We conclude with numerical simulations of the coupled two dimensional model with periodic boundary conditions.

2:54PM D18.00004 Modeling Cell Desiccation with Glass Formation[1] , CHRIS VOGL, MICHAEL MIKSIS, STEPHEN DAVIS, Northwestern University, DAVID SALAC, University at Buffalo SUNY — Lyopreservation is a preservation technique that seeks to store cells at room temperature. The storage process involves desiccating cells filled with special glass-forming sugars. However, care must be taken during the desiccation process to avoid both damaging membrane stress and to ensure the complete formation of glass throughout the cell. A lipid vesicle model is used here to study this process. The vesicle is represented as a moving interface (tracked by a level set method) with bending rigidity and incompressibility conditions. The vesicle is placed in a fluid containing a spatially varying sugar concentration field. As physical cells are impermeable to the sugar, the vesicle's impermeability is enforced with an immersed interface method. This results in a concentration jump across the vesicle's membrane, which in turn determines the flow of fluid through the membrane. Various desiccation techniques are simulated with this model. The effectiveness of each technique at minimizing membrane stress while ensuring glass formation will be discussed.

[1]We acknowledge partial financial support from NIH and RTG grants.

3:07PM D18.00005 ABSTRACT WITHDRAWN –

3:20PM D18.00006 The influence of cell crawling onto cell–cell chemical signaling , ROLAND BOUFFANAIS, Singapore University of Technology and Design — Chemotactic cells such as amoebae and leukocytes are able to aggregate and self-organize by means of local cell–cell chemical signaling. The chemical cAMP, which is produced by the cell, diffuses through the fluid from the emitting cell's membrane and binds to the neighboring cells' chemoreceptors. Such a purely diffusive view of this chemical signaling process fails to account for the fact that the cell's membrane constantly underges motions in relation with the specific motile behavior of these cells, namely crawling. We investigate the influence of cell motion/crawling onto the effectiveness of short-range chemical signaling. Our model is built on the study of an advection-diffusion process at the microscale of a cell for which diffusion is relatively "fast," and the flow generated by the cell while crawling is an incompressible Stokes flow given the smallness of the Reynolds number. A particular emphasis is placed on the effects of advection onto the generation of a steeper chemical gradient which can have a significant impact onto the chemosensing effectiveness.

3:33PM D18.00007 Multiscale Modeling of Virus Entry via Receptor-Mediated Endocytosis , JIN LIU, School of Mechanical and Materials Engineering, Washington State University — Virus infections are ubiquitous and remain major threats to human health worldwide. Viruses are intracellular parasites and must enter host cells to initiate infection. Receptor-mediated endocytosis is the most common entry pathway taken by viruses, the whole process is highly complex and dictated by various events, such as virus motions, membrane deformations, receptor diffusion and ligand-receptor reactions, occurring at multiple length and time scales. We develop a multiscale model for virus entry through receptor-mediated endocytosis. The binding of virus to cell surface is based on a mesoscale three dimensional stochastic adhesion model, the internalization (endocytosis) of virus and cellular membrane deformation is based on the discretization of Helfrich Hamiltonian in a curvilinear space using Monte Carlo method. The multiscale model is based on the combination of these two models. We will implement this model to study the herpes simplex virus entry into B78 cells and compare the model predictions with experimental measurements.

3:46PM D18.00008 Interaction between endothelial cells and albumin encapsulated droplets in Poiseuille flow[1] , ROBINSON SEDA[2], J. BRIAN FOWLKES[3], JOSEPH BULL[4], University of Michigan - Ann Arbor — Acoustic droplet vaporization (ADV) of DDFP encapsulated microdroplets has the ability to transform these emulsions into larger gas emboli capable of occluding blood vessels for therapy. An albumin shell is able to stabilize the droplet's superheated core, but can also interact with endothelial cells (EC) at the vessel wall if in close proximity. Radial migration of these microdroplets could bring them close enough to make this interaction possible leading to bioeffects that include cell detachment and death if an ADV event occurs. The purpose of this study is to investigate the hydrodynamic conditions (i.e. shear stresses) that make possible this EC-droplet interaction. A flow chamber coated with a monolayer of EC and connected to a syringe pump is used to flow a DDFP droplet solution at physiological shear stresses (1-50 dyne/cm^2) and inspected for droplet attachment. Droplets have been observed to interact and reversibly attach to EC in a static environment, thus it is expected that at low shear stress values interaction and further attachment will be possible. Knowing the flow conditions at which this interaction is likely to occur will aid in preventative measures to avoid significant bioeffects associated with ADV near the vessel wall.

[1]This work is supported by NIH grant R01EB006476.

[2]Department of Biomedical Engineering

[3]Department of Radiology

[4]Department of Biomedical Engineering

3:59PM D18.00009 Modeling Lymphoma Growth in an Evolving Lymph Node Using a Diffuse Domain Approach[1] , YAO-LI CHUANG, University of New Mexico (Pathology), VITTORIO CRISTINI, University of New Mexico (Pathology, Chemical Engineering), YING CHEN, XIANGRONG LI, University of California, Irvine (Mathematics), HERMANN FRIEBOES, University of Louisville (Bioengineering), JOHN LOWENGRUB, University of California, Irvine (Mathematics, Biomedical Engineering, Chemical Engineering and Material Science) — Tumor growth often poses as a multiphase free-boundary problem as tumor cells aggregate into distinct subdomains due to differentiated cell-cell and cell-matrix adhesion. In "Three-dimensional multispecies nonlinear tumor growth - I Model and numerical method" [Wise et al., J. Theor. Biol. 253, pp. 524-543 (2008)], we have developed a multiphase Cahn-Hilliard model to study morphological patterns of tumor growth in a homogeneous open environment, and the results resembled in-vitro experiments. In living tissues, however, tumors are often confined in a closed environment of an organ, where the tissue geometry can also evolve in response to the pressure of tumor growth. Here we adapt our previous Cahn-Hilliard tumor growth model to an evolving geometry using a recently developed diffuse domain approach. We use the model to study the growth of lymphoma in a lymph node that swells during the process. An angiogenesis model for tumor-induced vasculature is also adapted to investigate substrate distribution and drug delivery within the lymph node.

[1]Supported by NIH-PSOC grant 1U54CA143907-01.

4:12PM D18.00010 Curling dynamics of naturally curved ribbons from high to low Reynolds numbers[1] , OCTAVIO ALBARRAN ARRIAGADA, GLADYS MASSIERA, MANOUK ABKARIAN, Laboratoire Charles Coulomb Université Montpellier 2-CNRS — Curling deformation of thin elastic sheets appears in numerous structures in nature, such as membranes of red blood cells, epithelial tissues or green algae colonies to cite just a few examples. However, despite its ubiquity, the dynamics of curling propagation in a naturally curved material remains still poorly investigated. Here, we present a coupled experimental and theoretical study of the dynamical curling deformation of naturally curved ribbons. Using thermoplastic and metallic ribbons molded on cylinders of different radii, we tune separately the natural curvature and the geometry to study curling dynamics in air, water and in viscous oils, thus spanning a wide range of Reynolds numbers. Our theoretical and experimental approaches separate the role of elasticity, gravity and hydrodynamic dissipation from inertia and emphasize the fundamental differences between the curling of a naturally curved ribbon and a rod described by the classical Elastica. Our work shows evidence for the propagation of a single instability front, selected by a local buckling condition. We show that depending on gravity, and both the Reynolds and the Cauchy numbers, the curling speed and shape are modified by the large scale drag and the local lubrication forces.

[1]This work was supported by the French Ministry of Research, the CNRS Physics-Chemistry-Biology Interdisciplinary Pro- gram, the University Montpellier 2 Interdisciplinary Program and the Region Languedoc-Roussillon.

Sunday, November 18, 2012 2:15PM - 4:25PM –

Session D19 Surface Tension I 28E - Chair: Satish Kumar, University of Minnesota

2:15PM D19.00001 Liquid Contact Line on Visco-Elastic Gels: Stick-Slip vs Continuous Motions , TADASHI KAJIYA, PPMD-SIMM, UMR 7615 CNRS, UPMC, ESPCIParisTech, ADRIAN DAERR, MSC, UMR 7057 CNRS, Universite Paris Diderot, TETSUHARU NARITA, PPMD-SIMM, UMR 7615 CNRS, UPMC, ESPCIParisTech, LAURENT ROYON, MSC, UMR 7057 CNRS, Universite Paris Diderot, FRANCOIS LEQUEUX, PPMD-SIMM, UMR 7615 CNRS, UPMC, ESPCIParisTech, LAURENT LIMAT, MSC, UMR 7057 CNRS, Universite Paris Diderot — We studied the dynamics of water sessile drops being inflated on hydrophobic and visco-elastic Poly(styrene-butadiene-styrene)-paraffin gel substrates. While advancing, the droplet contact line exhibits three different motions. When the contact line advances at a high velocity, the contact line moves continuously with a constant contact angle. As the contact line slows down, it starts stick-slip: the contact line stays at a certain position then suddenly slips forward. With further decrease of the velocity, the contact line stops stick-slip and continuously advances again. The type of motions (continuous - stick-slip - continuous) depends on drop radius and mean velocity only through their ratio, $f = v/R$, which is a typical frequency of the contact line motion. The observed transitions of the contact line motions indicate that the rheology of gel drastically contributes to the wetting dynamics on its surface. On visco-elastic gels, the moving contact line exhibits both aspects of wetting on elastic solids and wetting on viscous liquids depending on f. At an intermediate regime, the stick-slip motion appears. In this conference, we will present our latest experimental results and suggest possible mechanisms explaining the present phenomena.

2:28PM D19.00002 Self-cleaning of superhydrophobic surfaces by spontaneously jumping condensate drops , KATRINA WISDOM, Duke University, JOLANTA WATSON, GREGORY WATSON, James Cook University, CHUAN-HUA CHEN, Duke University — The self-cleaning function of superhydrophobic surfaces is conventionally attributed to the removal of contaminating particles by impacting or rolling water droplets, which implies the action of external forces such as gravity. Here, we demonstrate a new self-cleaning mechanism, whereby condensate drops spontaneously jump upon coalescence on a superhydrophobic surface, and the merged drop self-propels away from the surface along with the contaminants. The jumping-condensate mechanism is shown to autonomously clean superhydrophobic cicada wings, where the contaminating particles cannot be removed by external wind flow. Our findings offer new insights for the development of self-cleaning materials.

2:41PM D19.00003 Interaction between counterpropagating Rossby and capillarity waves in planar jets and wakes , LUCA BIANCOFIORE, Imperial College London, FRANCOIS GALLAIRE, EPFL, PATRICE LAURE, University of Nice - Sophia Antipolis — By means of a global linear analysis, Tammisola et al. (2011) have observed a counterintuitive destabilizing effect of the surface tension in planar wakes. They have justified this destabilization by the presence of two different temporal unstable modes found when analyzing the local stability of an extracted velocity profile from the base flow. In the present study, we approximate the velocity profile of a jet/wake flow through a piecewise broken-line. We then explain the presence of these two temporal unstable modes for such flows using the counterpropagating Rossby wave (CRW) perspective (see Heifetz et al., 1999), which associates to each vorticity discontinuity an individual Rossby wave. The introduction of a finite amount of surface tension at the interface creates two capillarity waves (CW) which move with the same velocity but in opposite directions. The interaction of this four waves originates the two temporal unstable modes for both sinuous and varicose symmetries. Analyses of the influence of the shear layer thickness δ_w and the confinement h on the behaviour of both CRWs and CWs and on their interaction are provided. Finally, comparisons to direct numerical simulations of jets/wakes including surface tension will complete the study.

2:54PM D19.00004 The dynamics of three-dimensional liquid bridges with pinned and moving contact lines , SATISH KUMAR, SHAWN DODDS, University of Minnesota, MARCIO CARVALHO, PUC-Rio (Brazil) — Liquid bridges with moving contact lines are relevant in a variety of natural and industrial settings, ranging from printing processes to the feeding of birds. While it is often assumed that the liquid bridge is two-dimensional, three-dimensional effects are prominent in many applications. To investigate this we solve Stokes equations using the finite element method for the stretching of a three-dimensional liquid bridge between two flat surfaces, one stationary and one moving. We find that whereas a shearing motion does not alter the distribution of liquid between the two plates, rotation leads to an increase in the amount of liquid resting on the stationary plate as breakup is approached. This suggests that a relative rotation of one surface can be used to improve liquid transfer to the other surface. We then consider the extension of non-cylindrical bridges with moving contact lines. We find that dynamic wetting, characterized through a contact line friction parameter, plays a key role in preventing the contact line from deviating significantly from its original shape as breakup is approached. By adjusting the friction on both plates it is possible to drastically improve the amount of liquid transferred to one surface while maintaining fidelity of the liquid pattern.

3:07PM D19.00005 Onset of Dynamic Wetting Failure: The Influence of the Displaced Fluid , ERIC VANDRE, University of Minnesota, MARCIO CARVALHO, Pontifícia Universidade Católica do Rio de Janeiro, SATISH KUMAR, University of Minnesota — Dynamic wetting involves the displacement of fluid on a solid surface by an advancing liquid, and is essential to the successful operation of coating processes. In this work, we consider a model problem in order to examine the influence of the displaced fluid on the failure of dynamic wetting. Full two-dimensional (2D) calculations over a broad range of parameters are performed using the finite element method (FEM), and the results are compared to prior experiments and asymptotic analysis. This comparison motivates the development of a novel and efficient hybrid computational method that combines 2D FEM for the liquid and lubrication theory for the displaced fluid. We will discuss the limits of applicability of the hybrid approach, and its ability to describe realistic coating flows. Overall, our results highlight the significant influence of the displaced fluid on the conditions at which dynamic wetting failure occurs, along with the underlying physical mechanisms.

3:20PM D19.00006 Pinching of a liquid ligament under surface tension , JEROME HOEPFFNER, Université Pierre et Marie Curie — We study the retraction of a cylinder of fluid caused by surface tension. At the tip of the ligament, a blob if formed which collect progressively the liquid as it retracts. Between the blob and the cylinder, there is the creation of a neck, whose radius decreases progressively following a mechanism close to that of the Rayleigh-Plateau instability. Inside this neck, we observe a jet of the fluid from the cylinder into the blob, a "capillary Venturi." As the radius of the neck decreases the intensity of this jet increases, and we observe by means of numerical simulations and experiments that the detachment of this jet and creation of a vortex ring is able to alter significantly the evolution of the pinching: the pinching is avoided. This phenomenon is significant in the context of atomization because it changes significantly the statistics of the drops which are created from the retraction of the liquid ligament.

3:33PM D19.00007 Physics, mathematics and numerics of particle adsorption on fluid interfaces , MARKUS SCHMUCK, Department of Chemical Engineering, Imperial College London, GRIGORIOS A. PAVLIOTIS, Department of Mathematics, Imperial College London, SERAFIM KALLIADASIS, Department of Chemical Engineering, Imperial College London — We study two arbitrary immiscible fuids where one phase contains small particles of the size of the interface and smaller. We primarily focus on charge-free particles with wetting characteristics described by the contact angle formed at the interface between the two phases and the particles. Based on the experimental observation that particles are adsorbed on the interface to reduce the interfacial energy and hence the surface tension as well, we formulate a free-energy functional that accounts for these physical effects. Using elements from calculus of variations and formal gradient flow theory, we derive partial differential equations describing the location of the interface and the density of the particles in the fluid phases. Via numerical experiments we analyse the time evolution of the surface tension, the particle concentration, and the free energy over time and reflect basic experimentally observed phenomena.

3:46PM D19.00008 Double-diffusive Marangoni convection around exothermic chemical fronts , L. RONGY, P. ASSEMAT, A. DE WIT, Nonlinear Physical Chemistry Unit, Universite Libre de Bruxelles (ULB), Brussels, Belgium — We study double diffusive Marangoni flows triggered by concentration and temperature gradients across an exothermic chemical front propagating in horizontal uncovered solution layers. We numerically integrate the incompressible Navier-Stokes equations coupled through the tangential stress balance to evolution equations for the concentration of the autocatalytic product and the temperature. Solutal and thermal Marangoni numbers quantify the effect of the concentration and temperature gradients on the surface tension respectively, while the Lewis number measures the ratio of thermal diffusivity over molecular diffusivity. The asymptotic isothermal dynamics is characterized by a steady fluid vortex traveling at a constant speed with the reactive front, deforming and accelerating it. We analyze here the influence of thermal effects on the dynamics of the system in both cases of cooperative and competitive solutal and thermal effects. In particular, because heat and mass diffuse at different rates, new unsteady double-diffusive dynamics such as oscillations of the concentration field can be observed when the solutal and thermal effects act antagonistically on the surface tension. The influence of the various parameters on the flow field is investigated.

3:59PM D19.00009 Optically controlled Marangoni Tweezers , STEFFEN HARDT, SUBRAMANYAN NAMBOODIRI, SAJAN GEORGE, TOBIAS BAIER, Center of Smart Interfaces, TU Darmstadt, Petersenstraße 32, 64287 Darmstadt, Germany, MARTINA EWALD, MARKUS BIESALSKI, Department of Chemistry, TU Darmstadt, Petersenstraße 22, 64287 Darmstadt, Germany — A novel method for trapping and manipulating small particles is reported. The method relies on photoresponsive surfactants adsorbed to a gas-liquid interface that can be reversibly switched between two isomeric states using a focused laser beam. The principle is based on local changes of the surface tension, giving rise to Marangoni stresses. Depending on the type of surfactant isomer in the region around the laser spot, a flow either radially inward or outward is created. It is studied how the flow field generated depends on the light intensity and on the surfactant concentration. It is shown how the optically-induced inward flow can be utilized to trap and manipulate microspheres adsorbed to the gas-liquid interface. This principle of optically-controlled Marangoni tweezers opens new application perspectives, for example for the manipulation of nanoparticles. This is due to the fact that hydrodynamic stresses exhibit a more favorable scaling with the particle size than the Maxwell stresses utilized in conventional optical tweezers.

4:12PM D19.00010 Harnessing Compositional Marangoni Flows in Depositing Nanoparticle Films , MAINAK MAJUMDER, Monash University, Clayton, Victoria, Australia, MATTEO PASQUALI, Rice University, Houston, Texas, USA, MONASH UNIVERSITY/RICE UNIVERSITY TEAM — Attempts at depositing uniform films of nanoparticles by drop-drying have been frustrated by the "coffee-stain" effect, arising from the convective macroscopic flow into the solid-liquid-vapor contact line of a droplet. We have recently demonstrated that uniform deposition of nanoparticles from aqueous suspensions can be obtained by drying the droplet in an ethanol vapor atmosphere .(.).....(Majumder et al., 2012). This technique allows the particle-laden water droplets to spread on a variety of surfaces such as glass, silicon, mica, PDMS, and even Teflon® due to absorption of ethanol from the vapor. Visualization of droplet shape and internal flow shows initial droplet spreading and strong re-circulating flow during spreading and shrinkage. During the drying phase, the vapor is saturated in ethanol, leading to preferential evaporation of water at the contact line; thereby generating a surface tension gradient (or Marangoni forces) that drive a strong recirculating flow. We show that this method can be used for depositing catalyst nanoparticles for the growth of single-walled carbon nanotubes as well as to manufacture plasmonic films of well-spaced, unaggregated gold nanoparticles. .MAJUMDER, M., RENDALL, C. S., PASQUALI, M. et al. 2012. Overcoming the "Coffee-Stain" Effect by Compositional Marangoni-Flow-Assisted Drop-Drying. *J.Phys.Chem.B,* 116, 6536-6542.

Sunday, November 18, 2012 2:15PM - 4:25PM –

Session D20 Turbulent Boundary Layers II: Structures 30A - Chair: Philipp Schlatter, KTH

2:15PM D20.00001 Time-resolved evolution of coherent structures in turbulent channels[1] , ADRIAN LOZANO-DURAN, Universidad Politecnica de Madrid, JAVIER JIMENEZ, Universidad Politecnica de Madrid, CTR - Stanford University — The temporal evolution of vortex clusters and of the structures responsible for the momentum transfer in turbulent channels at $Re_\tau = 950$, 2000 and 4000 are studied using DNS sequences with temporal separations among fields short enough for individual structures to be tracked. From the geometric intersection of structures in consecutive fields we build temporal connection graphs of all the objects and define main and secondary branches in a way that each branch represents the temporal evolution of one coherent structure. A family of evolutions is found with self-similar sizes and lifetimes that can be born at any height with respect to the wall, although the probability increases close to it. Especial attention is paid to the wall-normal displacement of the structures. Sweeps tend to go towards the wall whereas ejections move away from it. In all the cases, the vertical velocity is close to u_τ and the wall-normal displacement is proportional to the lifetime of the structures and to their sizes. Finally, direct and inverse physical cascades are defined, associated with the process of splitting and merging among structures. The direct cascade predominates, but both directions are roughly comparable.

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

2:28PM D20.00002 On the near-wall vortical structures at high Reynolds numbers , PHILIPP SCHLATTER, RAMIS ORLU, QIANG LI, Linne FLOW Centre, KTH Mechanics, FAZLE HUSSAIN, University of Houston, DAN HENNINGSON, Linne FLOW Centre, KTH Mechanics — A recent database from direct numerical simulation (DNS) of a turbulent boundary layer (TBL) up to $Re_\theta = 4300$ has been analysed to educe the dominant flow structures in the near-wall region. In particular, the question of whether hairpin vortices are significant features of TBL is addressed. It is shown that during the initial phase of laminar-turbulent transition induced via tripping, hairpin vortices evolving from transitional vortices are numerous, and can certainly be considered as the dominant structure of the immediate post-transition stage of a boundary layer. This is in agreement with previous experiments and various low Reynolds-number simulations. At sufficient distances farther downstream from transition, the flow is dominated by a staggered array of quasi-streamwise vortices which are the same as observed in channel flow studies. It turns out that even quantitatively, no major differences between boundary layers and channels can be detected. The present results confirm that there is i) no reason why transitional hairpin vortices should persist in the fully developed turbulent region, ii) that the regeneration process does not involve hairpin vortices, and iii) that their dominant appearance as instantaneous outer structures is unlikely.

2:41PM D20.00003 Merging and auto-generation of vortices in wall bounded flow , GERRIT ELSINGA, MANU GOUDAR VISHWANATHAPPA, WIM-PAUL BREUGEM, Delft University of Technology — For channel flow, we explore how a hairpin eddy may reach a threshold strength required to produce additional hairpins by means of auto-generation. This is done by studying the evolution of two eddies with different initial strengths (but both below the threshold strength), initial sizes and initial stream-wise spacing between them. The numerical procedure followed is similar to Zhou et al [1]. The two eddies were found to merge into a single stronger eddy in case of a larger upstream and a smaller downstream eddy placed within a certain initial stream-wise separation distance. Subsequently, the resulting stronger eddy was observed to auto-generate new eddies. Merging of eddies thus is a viable explanation for the creation of the threshold strength eddies.

[1] J. Zhou, R.J. Adrian, S. Balachandar, and T.M. Kendall, Journal of Fluid Mechanics, 387:353-396, 1999.

2:54PM D20.00004 ABSTRACT WITHDRAWN –

3:07PM D20.00005 Application of vortex identification schemes to DNS data of transitional boundary layer[1] , BRIAN PIERCE, PARVIZ MOIN, TARANEH SAYADI, Stanford University, Center for Turbulence Research — We have demonstrated how various vortex identification and visualization criteria perform using DNS data from a transitional and turbulent boundary layer by Sayadi et al. (submitted to J. Fluid Mech.). The presence of well-known Λ vortices in the transitional region provides a well defined and yet realistic benchmark for evaluation of various criteria. We investigate the impact of changing the threshold used for iso-surface plotting.

[1]Brian Pierce acknowledges the support from the Stanford Graduate Fellowship.

3:20PM D20.00006 Development and Interaction of Artificially Generated Hairpin Vortices[1] , DANIEL SABATINO, CHRISTOPHER MCKENNA[2], Lafayette College — The development and interaction of hairpin vortices are examined and categorized to better understand their role in fully turbulent boundary layers. Hairpin vortices are generated within an otherwise laminar boundary layer using a free surface water channel. Direct injection is the primary generation method and the behavior of the vortices is first examined using flow visualization. Hydrogen bubble wire is combined with dye injection to help clarify the role of the vorticity in the fluid immediately surrounding the hairpin vortex. PIV data is also used to classify the development and maturity of the vortices for a range of free stream and injection conditions. The interactions of two hairpin vortices of varying maturity are characterized to investigate the potential mechanisms for the formation of hairpin packets beyond autogeneration. Finally, the behavior of hairpin vortices generated with a new technique that uses a transient hemispherical protrusion is also examined.

[1]Supported by the National Science Foundation under Grant CBET-1040236

[2]Now at Lehigh University

3:33PM D20.00007 Two-point correlations for zero-pressure-gradient turbulent boundary layers and channels at $Re_\tau \approx 1000 - 2000$[1] , JUAN A. SILLERO, JAVIER JIMÉNEZ, U. Politécnica de Madrid, ROBERT D. MOSER, U. Texas at Austin — Two-point 5-dimensional correlations $C_{\xi\xi}(x; x'; y; y'; \Delta z)$ are investigated to educe the structure of the velocity and pressure fluctuations in zero-pressure-gradient turbulent boundary layers in the range $Re_\theta = 2780 - 6680$, and in matching channels at $Re_\tau \approx 1000 - 2000$. Eddies in channels are coherent over longer distances than in boundary layers, especially for C_{uu} in the direction of the flow. At the 5% level, the maximum streamwise length of C_{uu} is $O(6\delta)$ for boundary layers and $O(15h)$ for channels. The corresponding lengths for the transverse velocities and for the pressure are shorter, $O(\delta$-$2\delta)$, and of the same order for both flows. Integral correlation lengths in the streamwise and spanwise directions grow away from the wall, except for $L_{uu,x}$, which peaks at $y \approx 0.6h$ in channels and at $y \approx 0.2\delta$ in boundary layers, probably due to the outer intermittency in the latter. Above the buffer layer, C_{uu} is inclined by $\approx 10 - 12^o$ from the wall, the wall-normal velocity and the pressure are roughly vertical, and C_{ww} is inclined by $\approx 30^o$. Those features seem unaffected by the Reynolds number and by the type of flow.

[1]Funded by CICYT, INCITE and ERC.

3:46PM D20.00008 Near-wall turbulent fluctuation in the absence of the wide outer motions , YONGYUN HWANG, DAMTP, University of Cambridge — Numerical experiments which filter out turbulent motions wider than $\lambda_z^+ \simeq 100$ are carried out up to $Re_\tau = 660$ in a turbulent channel. The mean-velocity profile shows very good agreement with that of full simulation below $y^+ \simeq 40$. The turbulent fluctuations mainly consist of two parts: the long streamwise velocity fluctuation confined to the near-wall region with $\lambda_x^+ \simeq 600 \sim 800$, and the short fluctuation of all the velocity components with $\lambda_x^+ \simeq 200 \sim 300$ inducing turbulence even above $y^+ \simeq 100$. In the absence of the wide outer motions, the former remains almost unchanged with the Reynolds number, resulting in an almost constant value of the near-wall maximum streamwise velocity fluctuation at all the Reynolds numbers considered. On the other hand, the latter strongly interacts with the non-realistic mean shear in the outer region and induces non-negligible amounts of the fluctuation near the channel center even at $Re_\tau = 660$. The removal of the wider structures also reveals significant amounts of drag reduction particularly at large Reynolds numbers, implying that the filtered wide outer structures are involved in generating turbulent skin friction.

3:59PM D20.00009 A comparison of spanwise vorticity fluctuations statistics over $\delta^+ = 350$ - 8000 , CALEB MORRILL-WINTER, RACHEL EBNER, University of New Hampshire, RIO BAIDYA, University of Melbourne, PETAR VUKOSLAVCEVIC, University of Montenegro, JOSEPH KLEWICKI, University of New Hampshire, University of Melbourne, JAMES WALLACE, University of Maryland, IVAN MARUSIC, University of Melbourne — The behaviors of the spanwise vorticity fluctuations (ω_z) and their correlation with the wall normal and streamwise velocity fluctuations are central to describing the momentum and kinetic energy transport mechanisms in the turbulent boundary layer. To date, however, well-resolved laboratory measurements of ω_z have been confined to low Reynolds numbers. Compact four-element hot-wire probe measurements are used to explore the statistical structure of the spanwise vorticity fluctuations in zero pressure gradient turbulent boundary layers. Existing well-resolved laboratory data, $\delta^+ = 375$, 970 & 1500, along with recently acquired data from the University of New Hampshire's low-speed boundary layer wind tunnel and University of Melbourne's High Reynolds Number Boundary Layer Wind Tunnel are examined. In the present flows, the spatial resolution of the probe ranged from $l^+ = 4$ - 12. The properties of individual velocity gradient contributions to ω_z are examined, along with the Reynolds number scaling behaviors of the first four statistical moments of ω_z. The present results indicate that the motions bearing spanwise vorticity exhibit a significant Reynolds number dependence in the wake region of the boundary layer.

4:12PM D20.00010 Small-scale statistics in direct numerical simulation of turbulent channel flow up to $Re_\tau = 5120$, KOJI MORISHITA, Kobe University, JST CREST, TAKASHI ISHIHARA, Nagoya University, JST CREST, YUKIO KANEDA, Aichi Institute of Technology — Small-scale statistics in high-Reynolds number turbulent channel flow are studied by direct numerical simulation (DNS) with Re_τ up to 5120. The DNS exhibits a range where the mean stream-wise velocity U fits well to the log-law and the energy dissipation rate is nearly inversely proportional to the distance from the wall. The width of the range increases with the Reynolds number. In the range, the spectrum $E_{12}(k_1)$ of the cross correlation between the stream-wise and span-wise fluctuating velocity components fits well to the -7/3 power law in the inertial sub-range, where k_1 is the wave number in the stream wise direction. Its pre-factor is in good agreement with laboratory experiments of turbulent boundary layer by Saddoughi et al. (1994) and Tsuji (2003), and DNS of homogeneous turbulent shear flow by Ishihara et al.(2002).

Sunday, November 18, 2012 2:15PM - 4:25PM –

Session D21 Turbulence Simulation: Isotropic/Homogenous Shear 30B - Chair: James Riley, University of Washington

2:15PM D21.00001 On the kinematics of scalar iso-surfaces in turbulent flow[1] , WEIRONG WANG, JAMES J. RILEY, JOHN C. KRAMLICH, University of Washington — The behavior of scalar iso-surfaces in turbulent flows is of fundamental interest and also of importance in certain applications, e.g., the stoichiometric surface in nonpremixed, turbulent reacting flows. Of particular interest is the average area per unit volume of the surface, Σ. We report on the use of direct numerical simulations to directly compute Σ and to model its evolution in time for the case of isotropic turbulence. Using both a direct measurement technique, and also Corrsin's (1955) suggestion of surface-crossing, we find the iso-surface in space and also measure Σ as the surface evolves in time. This allows us to follow the growth of the surface due to local surface stretching and its ultimate decrease due to molecular destruction. We are also able to measure the principal terms in the evolution equation for Σ, including the surface stretching term $\mathcal{S}$ and the molecular destruction term $\mathcal{M}$. For example, for the scalar Z we find that its spatial derivative quantities are approximately statistically independent of Z itself, so that $\mathcal{S}$ and $\mathcal{M}$ are approximately statistically independent of Z as well. Finally, a model is proposed which fairly accurately predicts the evolution of Σ.

[1]Supported by NSF Grant No. OCI-0749200.

2:28PM D21.00002 Local topology of energy transport in isotropic turbulence[1] , JONAS BOSCHUNG[2], CHARLES MENEVEAU, Johns Hopkins University — Similar to the velocity vector field, whose tangent (stream) lines represent how fluid volume (or mass in constant density flows) is transported in the flow, it is of interest to consider the vector field corresponding to the transport of mechanical energy (Meyers & Meneveau, 2012). The transport includes advection and viscous diffusion. In order to characterize the local topology of this vector field in turbulence, we examine statistical properties of its gradient field. This energy transport field is not divergence-free, due to dissipation and unsteady changes of kinetic energy. Therefore, the first invariant (the trace) of its gradient tensor is not zero, as in compressible flow. The three invariants P_E, Q_E and R_E of the energy transport gradient tensor are analyzed using concepts developed earlier for analysis of compressible flows. Data from DNS of isotropic turbulence is used, from the JHU database (Li et al. 2008, JoT), as well as other sources. Contracting node-like topology occurs very frequently, consistent with the dissipative nature of fluid turbulence. Further topological properties are established based on conditional PDFs of the invariants, and flow visualizations are used to develop insights into the local structure of the energy transport vector field

[1]This work is supported by project CMMI-0941530. The authors also thank Prof. J. Meyers, Prof. N. Peters and Mr. P. Schaefer for interesting discussions on this topic.

[2]Permanent address: Department of Mechanical Engineering, RWTH Aachen, Germany.

2:41PM D21.00003 Scaling of Lyapunov Exponents in Homogeneous, Isotropic DNS , NICHOLAS FITZSIMMONS, MYOUNGKYU LEE, NICHOLAS MALAYA, ROBERT MOSER, University of Texas Austin — In order to study the nature of the chaos in turbulence, we investigate Lyapunov exponents. These exponents measure the rate of separation of initially infinitesimally close trajectories in phase space. Lyapunov exponents are examined for two purposes: to investigate the scaling of the exponents with respect to the parameters of forced homogeneous isotropic turbulence, and to locate the chaotic features of turbulence. Specifically, we explore the scaling of the Lyapunov exponents with respect to the Taylor Reynolds number, Re_λ, and with respect to the ratio of the integral length scale and the computational domain. The latter is varied through manipulation of the Uhlenbeck Ornstein process, which forces the DNS. The exponents are measured by introducing a linear disturbance, evolving it with the linearized Navier-Stokes equation, and normalizing it at each step. Using this disturbance and the velocity field one calculates the instantaneous growth rate of the disturbance at each time step. We will show how these exponents may then be used to measure the predictability in turbulent flows and allow for the study of the instabilities of the chaotic field.

2:54PM D21.00004 ABSTRACT MOVED TO M17.00010 –

3:07PM D21.00005 Inverse energy cascade in rotational turbulence , HUIDAN (WHITNEY) YU[1], College of Metrology & Measurement Engineering, Zhongguo Jiliang University, Hangzhou, China, ROU CHEN[2], Indiana University-Purdue University Indianapolis (IUPUI), HENGJIE WANG, Peking University, China — Rotation influences large-scale motions in the Earth's atmosphere and oceans and it is also important in many industrial applications such as turbo machinery, rotor-craft, and rotating channel etc. We study rotation effects on decaying isotropic turbulence through direct numerical simulation using lattice Boltzmann method. A Coriolis force characterized by the angular velocity of the frame of reference Ω is included in the lattice Boltzmann equations. The effects of rotation on fundamental turbulence features such as kinetic energy and enstrophy decay, energy spectrum, etc. are studied. The decay laws are quantitatively captured. Inverse energy cascade are observed in the 3D turbulence with and without rotation. The scaling of the inverse energy cascade and its relation to initial energy spectrum are explored.

[1]Indiana University-Purdue University Indianapolis (IUPUI)
[2]Zhejinag Normal University, China

3:20PM D21.00006 Simulation of Homogeneous Turbulence Subjected to Plane Strain[1] , CHRIS ZUSI, J. BLAIR PEROT, University of Massachusetts, Amherst — Direct numerical simulation is used at a resolution of 512^3 to investigate the behavior of turbulence subjected to rotation, as well as plane and axi-symmetric strain. The initial isotropic turbulence is generated by the stirring action of many small randomly placed cubes, rather than imposed as an initial condition. Anisotropic turbulent structure is then generated by rotation, dimensionless plane strain or axi-symmetric strain. Multiple simulations are used to investigate the influence of initial conditions, rotation rate, strain rate and Reynolds number on the strained turbulence structure and its subsequent anisotropic decay.

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

3:33PM D21.00007 The effect of aspect ratio on statistically-stationary homogeneous shear flow[1] , SIWEI DONG, ATSUSHI SEKIMOTO, JAVIER JIMÉNEZ, Universidad Politécnica de Madrid — The effect of the aspect ratio of the numerical domain is investigated in homogeneous shear turbulence at moderate $Re_\lambda = 40$-100, over long times for which its length scales are constrained by the numerics. For $L_y/L_z > 1$, the cross-shear box size, L_y, has a negligible influence on the statistics. For $A = L_x/L_z \leq 2$, the flow contains a relatively steady streamwise-velocity (u) streak, and the r.m.s. u' dominates the energy. For $A \geq 2$, the integral length is proportional to the spanwise box size L_z and the r.m.s. velocities are proportional to SL_z. That regime is dominated by strong bursting. In $2 < A < 4$, the bursting is due to the quasi-periodic breakdown of the streaks, which results in the intermittent formation of strong quasi-streamwise vortices. In that range, parameters such as the dimensionless shear Sq^2/ϵ, and the bursting period $ST \approx 10$-15 are of the same order as in the logarithmic layer of wall turbulence. For $A \gg 4$, the cross-shear velocity v bursts most strongly, and the streak is weaker. That regime is quasi-two-dimensional in the (x, y) plane.

[1]Funded by ERC and the Chinese Scholarship Council

3:46PM D21.00008 Direct numerical simulations of statistically-stationary homogeneous shear turbulence[1] , ATSUSHI SEKIMOTO, SIWEI DONG, JAVIER JIMÉNEZ, U. Politecnica Madrid — The long-term behaviour of homogeneous shear turbulence is studied using a new direct simulation code. The incompressible Navier-Stokes and continuity equations are formulated in terms of the vertical vorticity and of the Laplacian of the vertical velocity. The domain is periodic in the streamwise (x) and spanwise (z) directions, and periodic between shifting points of the lower and upper y-boundaries. The discretization is dealiased Fourier in (x, z) and compact finite differences in y, with the shear-periodic boundary conditions embedded in the finite-difference matrices for each Fourier mode. There is no recurrent remeshing, thus avoiding the secular loss of enstrophy during long integration times. The code was validated using linear theory, as well as the initial shearing of isotropic turbulence. The results depend only weakly on the vertical box dimension. Over long times it develops the streaks and quasi-periodic bursting behaviour typical of wall-bounded turbulence, with mean shear parameters, Sq^2/ϵ, of the same order as those in the logarithmic layer of turbulent channels, suggesting that it can be used as a model for the self-sustaining mechanism of inertial wall turbulence.

[1]Funded by the ERC

3:59PM D21.00009 Turbulence generation through concentrated momentum sources[1] , AGUSTIN MAQUI, DIEGO DONZIS, Texas A&M University — In many applications, thermal non-equilibrium can be a major contributor to the dynamics of turbulence. One example amenable to laboratory experiments is that of laser induced photo-dissociation of molecules, which can create fragments with very large velocities. It is of fundamental and practical interest to investigate whether this large momentum excess can be used to generate realistic turbulence. Consequently, direct numerical simulations (DNS) with concentrated momentum sources that reproduce the photo-dissociation process have been conducted to study the creation and evolution of turbulence.Two critical times are found between which the flow quickly reorganizes into fully developed turbulence. The characteristic time scales are successfully scaled with parameters related to the initial conditions and the establishment of turbulence is studied through the evolution of the velocity gradients, spectra, and anisotropy measures. Preliminary results indicate that compressible turbulence is reached at an earlier stage than incompressible. Furthermore, it is shown that higher Taylor Reynolds numbers (R_λ) can be reached for compressible flows with weaker momentum sources. Further results and consequences for particular cases realizable in laboratories will be discussed.

[1]The authors gratefully acknowledge the support of AFOSR.

4:12PM D21.00010 Mechanism of axis-switching in low aspect-ratio rectangular jets , NAN CHEN, HUIDAN (WHITNEY) YU, Indiana University-Purdue University Indianapolis (IUPUI) — Axis-switching (AS) refers to the change in the orientation of the major axis of the jet from initial spanwise to later lateral direction. This phenomenon is of great interest both from fundamental physics and practical application points of view. AS is most noticeable in square and low aspect-ratio (AR) rectangular jet flows. It has been reported experimentally and computationally that square jet and rectangular jets switch the major axis 45^0 and 90^0 respectively. In this work we explore the mechanism of AS phenomenon through direct numerical simulation using lattice Boltzmann method for 5 rectangular jets with different aspect ratios at relatively low Reynolds numbers. We identified the three characteristic regions of jet flow that are potential core (PC), characteristic decay (CD), and axisymmetric decay (AD) regions. It is found that 45^0 (square jet) and 45^0 first then 90^0 (rectangular jets) AS occur in the CD region. The correlation between jet propagate velocity and coherent structure shows that the 45^0 AS occurs close to the entrance of CD region where the correlation gets the maximum value, indicating that the 45^0AS is driven mainly by the jet propagatation. The 90^0AS appears close to the end of the CD region and doesn't show unified relation to the correlation for different jets. The mechanism of 90^0AS is more complicated because both jet propagation and boundary condition contribute the driven. Meanwhile, flow pattern and vortices are closely looked into to reveal the mechanism of AS phenomena.

Sunday, November 18, 2012 2:15PM - 3:59PM –

Session D22 Turbulence Modeling: LES Wall Models 30C - Chair: Dale Pullin, California Institute of Technology

2:15PM D22.00001 LES of a turbulent channel flow with a predictive wall model[1] , MICHIO INOUE, DALE PULLIN, California Institute of Technology, IVAN MARUSIC, ROMAIN MATHIS, University of Melbourne — Large-eddy simulations (LES) of turbulent channel flow are presented. These LES combine the stretched-vortex, subgrid-scale (SGS) model with a near-wall model (Chung & Pullin JFM 2009) that here provides a plane-averaged, wall-shear stress together with a plane-averaged slip velocity at a raised or "virtual" wall. Instantaneous stream-wise fluctuations are then added to this slip velocity using an empirical inner-outer wall model (Mathis $et\ al$, JFM 2011) driven by a velocity time-series obtained from within the log-layer of the outer flow. This spatially fluctuating slip velocity is used as a boundary condition for the outer LES. Results using several variations of this wall model are described for turbulent channel flow at Reynolds numbers Re_τ up to 2.2×10^4.

[1]NSF CBET-1235605

2:28PM D22.00002 Turbulent boundary layer, wall shear-stress statistics using a predictive wall-model combined with LES[1] , DALE PULLIN, MICHIO INOUE, California Institute of Technology, ROMAIN MATHIS, IVAN MARUSIC, University of Melbourne — Time-wise velocity signals obtained from large-eddy simulation (LES) within the near-wall, logarithmic region of the zero-pressure gradient, flat-plate turbulent boundary layer are used as input to a calibrated, empirical wall model (Mathis $et\ al$, 2011) to calculate the statistics of the fluctuating, wall shear stress τ_w. These are compared with DNS (Schaltter & Örlü, 2011; Komminaho & Skote, 2002) at lower Reynolds number and with statistics obtained using the empirical wall model applied to experimentally generated time-series. The DNS, experimentally-based and LES-based predictions are consistent with a log-like increase of $\overline{(\tau_w'^+)^2}$ with Re_τ. It is argued that the LES is thus able to capture large-scale motions within the log-region that are generating this increased wall activity, up to $Re_\tau = 2 \times 10^5$.

[1]NSF CBET-1235605

2:41PM D22.00003 Total shear stress boundary condition at upper boundary of RANS in wall-modeled large eddy simulation[1] , CHANGWOO SUNG, JUNGIL LEE, HAECHEON CHOI, Seoul National University — In wall-modeled large eddy simulation, how to exchange the flow field information between the solutions from RANS and LES through the boundary condition is one of the important issues. In general, the wall boundary condition of LES is given as a form of the instantaneous wall shear stresses from the solution of RANS, whereas the upper boundary condition of RANS is provided as the instantaneous velocity from the solution of LES. However, in this approach, the total shear stress at the upper boundary is not continuous and thus momentum transfer from LES to RANS is not strictly conserved. In our study, we provide the instantaneous total shear stresses at the upper boundary of RANS with mixing-length model and conduct simulations of turbulent channel flow at high Reynolds numbers. The results show excellent predictions of turbulence statistics.

[1]Supported by the WCU and NRF programs

2:54PM D22.00004 A nested-LES wall-modelling approach for high Reynolds number wall-bounded turbulence , YIFENG TANG, RAYHANEH AKHAVAN, University of Michigan, Ann Arbor, MI 48109-2125 — A new wall-modelling approach for LES of high Reynolds number wall-bounded turbulence is proposed. The method couples coarse-grained LES in a full-size channel with nested fine-grained LES in a minimal channel. At each iteration, the fluctuating velocity field in both channels is rescaled to match the TKE components to that of the minimal channel in the near-wall region ($z^+ < 100$), to that of the full-size channel in the core ($z^+ > 300$), and to a weighted average of the two in between. Results were insensitive to the details and width of the weighting function. Simulations were performed for $1000 \leq Re_\tau \leq 10,000$ in full channels of size $2\pi h \times \pi h \times 2h$ and minimal channels of size $3000 \times 1500 \times 2Re_\tau$ wall units in the streamwise, spanwise and wall-normal directions, respectively. At all Re_τ, resolutions of $64 \times 64 \times 65$ in the full-size channel and $32 \times 64 \times 65$ in the minimal channel were employed, rendering the cost of computations independent of Re_τ. The Dynamic Smagorinsky model was used as the SGS model. The results show that the nested-LES approach can predict a friction coefficient within 5% of Dean's correlation, and one-point statistics in good agreement with available DNS and experimental data.

3:07PM D22.00005 Mean wall shear stress boundary condition for large eddy simulation with coarse mesh near the wall[1] , JUNGIL LEE, MINJEONG CHO, HAECHEON CHOI, Seoul National University — Mean wall shear stress is proposed for the wall boundary condition for large eddy simulation without resolving near-wall region. The motivation of using this wall boundary condition instead of no-slip boundary condition is that with very coarse resolution near the wall providing an accurate mean wall shear stress is most important in the momentum transport near the wall. As test problems, we consider two canonical wall-bounded flows at high Reynolds number: turbulent channel and boundary layer flows. First, the mean wall shear stress is obtained from the momentum balance for channel flow or from an empirical correlation of skin friction for boundary layer flow. The present boundary condition provides excellent predictions of the mean flow statistics, even if the first off-wall grid locates far away from the wall, $y^+ = O(10^1 \sim 10^3)$, where y is the wall-normal distance from the wall. Next, a dynamic approach based on the log-law is developed to obtain mean wall shear stress during computation and is applied to both flows, showing also excellent results.

[1]Supported by the WCU and NRF Programs

3:20PM D22.00006 Computing transitional flows using wall-modeled large eddy simulation , JULIEN BODART, JOHAN LARSSON, Center for Turbulence Research, Stanford University — To be applicable to complex aerodynamic flows at realistic Reynolds numbers, large eddy simulation (LES) must be combined with a model for the inner part of the boundary layer. Aerodynamic flows are, in general, sensitive to the location of boundary layer transition. While traditional LES can predict the transition location and process accurately, existing wall-modeled LES approaches can not. In the present work, the behavior of the wall-model is locally adapted using a sensor in the LES-resolved part of boundary layer. This sensor estimates whether the boundary layer is turbulent or not, in a way that does not rely on any homogeneous direction. The proposed method is validated on controlled transition scenarios on a flat plat boundary layer, and finally applied to the flow around a multi-element airfoil at realistic Reynolds number.

3:33PM D22.00007 Dynamic wall-modeling for LES of shock/boundary-layer interacting separated flow at high Reynolds number[1] , SOSHI KAWAI, Institute of Space and Astronautical Science, JAXA, JOHAN LARSSON, Center for Turbulence Research, Stanford University — We present a new dynamic procedure for non-equilibrium wall-modeling in large-eddy simulation (LES) at arbitrarily high Reynolds numbers. The proposed dynamic non-equilibrium wall-model is based on the methods that model the wall shear stress directly, and solves the full RANS equations in the wall-model layer. We first show how the existing non-equilibrium wall-model becomes inaccurate at high Reynolds number and then propose an improved method which solves this issue. The improvement stems directly from reasoning about how the turbulence length scale changes with wall distance in the inertial sublayer and the resolution-characteristics of numerical methods. The proposed method is shown to accurately predict both equilibrium boundary layers and non-equilibrium shock-induced separated boundary layer at very high Reynolds number, with both realistic instantaneous turbulent structures and accurate statistics (skin friction and turbulence quantities) without the use of ad hoc corrections, something that existing non-equilibrium wall-models fail to do robustly.

[1]This work was supported in part by the JAXA International Top Young Fellowship Program, Grant-in-Aid for Young Scientists (B) KAKENHI 24760670, and the US Air Force Turbulence Program (Grant FA9550-11-1-0111).

3:46PM D22.00008 Near-Wall Modeling for Large Eddy Simulation of Convective Heat Transfer in Turbulent Boundary Layers[1] , HYUN WOOK PARK, KIYOUNG MOON, Dept. CSE, Yonsei University, EZGI OZTEKIN, Technology & Management International, RANDALL MCDERMOTT, Fire Research Division, NIST, CHANGHOON LEE, JUNG-IL CHOI, Dept. CSE, Yonsei University — Necessity of the near-wall treatments for the large eddy simulation (LES) without resolving viscous layer is well known for providing a smooth transition from molecular to turbulent transport near wall region. We propose a simple but efficient approach based on modeling of wall shear stress and heat flux that enable accurate predictions of Nusselt number correlations for equilibrium boundary layers. The wall shear stress is directly modeled with Werner and Wengle (1991)'s power law model and wall heat flux is modeled with analogous wall laws between velocity and temperature with Kader (1981)'s empirical correlation. We perform the wall-modeled LES of turbulent convective heat transfer in a channel for various Prandtl numbers. The results show good agreement with the available experimental and numerical data.

[1]Supported by WCU (R31-10049) and EDISON (2012-0006663) program of NRF.

Sunday, November 18, 2012 2:15PM - 4:12PM –

Session D23 Turbulence Theory: General I 30D - Chair: Gregory Eyink, Johns Hopkins University

2:15PM D23.00001 Analytical Model for Pair Dispersion in Gaussian Models of Eulerian Turbulence , GREGORY EYINK, DAMIEN BENVENISTE, The Johns Hopkins University — Synthetic models of Eulerian turbulence are often used as computational shortcuts for studying Lagrangian properties of turbulence (e.g. Elliott & Majda, 1996). These models have been criticized by Thomson & Devenish (2005), who argued on physical grounds that their sweeping effects are very different from true turbulence. We give analytical results for Eulerian turbulence modeled by Gaussian fields. Our starting point is an exact integrodifferential equation for the particle pair separation distribution obtained from Gaussian integration-by-parts. When velocity correlation times are short, a Markovian approximation leads to a Richardson-type diffusion model. We obtain a time-dependent pair diffusivity tensor of the form $K_{ij}(\mathbf{r},t) = S_{ij}(\mathbf{r})\tau(r,t)$ where $S_{ij}(\mathbf{r})$ is the structure-function tensor and $\tau(r,t)$ is an effective correlation time of velocity increments. Crucially, this is found to be the minimum value of three times: the intrinsic turnover time $\tau_{eddy}(r)$ at separation r, the overall evolution time t, and the sweeping time r/v_0 with v_0 the rms velocity. We thus verify the main argument of Thomson & Devenish (2005), but we predict scaling laws for pair dispersion different from theirs for zero-mean velocity ensembles.

2:28PM D23.00002 Deviations from Kolmogorov-Kraichnan similarity theory in the energy cascade of two-dimensional alpha turbulence[1] , BEL HELEN BURGESS, University of Toronto, THEODORE SHEPHERD, University of Reading — We study energy cascades in 2D α turbulence, for which "vorticity" θ is related to streamfunction ψ by $\theta(\mathbf{x}) = (-\Delta)^{\alpha/2}\psi(\mathbf{x})$, where $(-\Delta)^{\alpha/2}$ is the fractional Laplacian. Using the eddy damped quasinormal Markovian (EDQNM) closure, we seek self-similar inertial range solutions. The energy flux is finite and the similarity solution self-consistent for $\alpha < 4$. In keeping with strain rate arguments, this suggests a spectrally local and self-similar energy cascade for $\alpha < 4$. However, the transfers vanish identically for $\alpha = 2.5$ and $\alpha = 10$. Comparison with statistical equilibrium spectra elucidates this: for $\alpha = 2.5$ and $\alpha = 10$, the similarity spectra coincide with enstrophy and energy equipartition respectively, and the similarity ranges are equilibrium solutions with Gaussian statistics. Moreover, the similarity range energy flux is toward small scales for $\alpha \in (2.5, 10)$, suggesting that any inverse cascade for $\alpha \geq 2.5$ cannot be self-similar. Numerical simulations confirm this: for $\alpha < 2.5$, one can obtain the similarity spectrum, while for $\alpha \geq 2.5$, the inverse cascade spectrum is significantly steeper than the similarity solution.

[1]B. H. Burgess was supported by an NSERC Canada Graduate Fellowship and an Amelia Earhart Fellowship from the Zonta International Foundation.

2:41PM D23.00003 On the decay of homogeneous nearly isotropic turbulence behind active fractal grids[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 appear to favor power-law decay with exponents between -1.2 and -1.4, approximately. More recently, fractal-generated turbulence (Hurst & Vassilicos, PoF 2007, and subsequent papers) using multi-scale passive grids suggest possible faster decay, and non-trivial behavior especially near the grid, where the mean velocity is spatially evolving. In order to generate spatially homogeneous flow using multi-scale injection of kinetic energy at high Reynolds numbers, we use a new type of active-grid consisting of winglets with various fractal shapes. We test space-filling fractal shaped winglets as well as Sierpisky-carpet and Apollonian packing type fractal shapes. Data are acquired using X-wire thermal anemometry. Tests of homogeneity of mean flow and turbulence intensity will be presented as well as decay of kinetic energy and spectral characteristics of the flow.

[1]This research is supported by NSF-CBET-1033942. The assistance of Ms. Imbi Salasoo and Mr. Nathan Greene in designing and building the fractal winglets is much appreciated. The authors also thank Mr. Vince Rolin for his assistance with the active grid.

2:54PM D23.00004 Can a flow be turbulent in microfluidics with Reynolds number in the order of 1? , GUIREN WANG, FANG YANG, WEI ZHAO, University of South Carolina — Traditionally, it is believed that turbulence occurs in relatively high Re number flow. For instance, the critical Re number is about 2100 in a pipe flow. Although there can be elastic turbulence in low Re, it is conventionally believed that the flow in mirofluidics, where typical Re is in the order of 1 or less, can only be laminar. Here, we demonstrate that features of turbulent flows can be achieved in a microchannel with Re in the order of 1, when the flow is electrokinetically forced. To measure the flow velocity, we developed a confocal microscopic velocimeter with high tempo-spatial resolution, i.e. molecular tracer based Laser Induced Fluorescence Photobleaching Velocimeter. We measured the general features in turbulent flows: fast diffusion or mixing, irregular flow velocity, high dissipation rate, 3-D flow and continuous power spectrum of velocity fluctuation indicating multiscale structures of small eddies. Interesting is that a -5/3 power spectrum with about one decade span in frequency is also observed. The results indicate that turbulence can be realized as well in microfluidics with Re in the order of 1. The study could open a new perspective view on turbulence and transport phenomena in microfluidics.

3:07PM D23.00005 On the collision of small particles in isotropic turbulence , SATOSHI YOKOJIMA, TAKASHI MASHIKO, KENJIRO BABA, TAKASHI MIYAHARA, Shizuoka University — Collisions of small particles in isotropic turbulence are closely investigated by direct numerical simulations. In the talk, the relationship between particle collision events and the background turbulent flow field will be discussed.

3:20PM D23.00006 Turbulent 2-Particle Dispersion Laws in Kinematic Simulations , DAMIEN BENVENISTE, GREGORY EYINK, Johns Hopkins University — Kinematic Simulations (KS) are often used as a shortcut for studying Lagrangian properties of turbulence (e.g. Elliott & Majda, 1996) but have been criticized by Thomson & Devenish (2005), who pointed out that KS sweeping effects are very different from true turbulence. We study numerically by a Monte Carlo method a Richardson-like diffusion equation recently derived analytically by us for KS models, which exhibits such sweeping effects. With moderate inertial-ranges like those achieved in current KS, our model is found to reproduce the $t^{9/2}$ power-law for pair dispersion predicted by Thomson & Devenish and observed in those KS. However, for much longer ranges, our model exhibits three distinct pair-dispersion laws in the inertial-range: a Batchelor t^2-regime, followed by a Kraichnan-model-like t^1 diffusive regime, and then a t^6 regime. Finally, outside the inertial-range, there is another t^1 regime with particles undergoing independent Taylor diffusion. These scalings are exactly the same as those predicted by Thomson & Devenish for KS with large mean velocities, which we argue hold also for KS with zero mean velocity. Our results support the basic conclusion of Thomson & Devenish (2005) that sweeping effects make Lagrangian properties of KS completely different from true turbulence for very extended inertial-ranges.

3:33PM D23.00007 Turbulence modulation through the interface of a deformable drop , LUCA SCARBOLO, University of Udine, DAFNE MOLIN, University of Brescia, ALFREDO SOLDATI, University of Udine — The transport of momentum across the interface of a large deformable droplet immersed in a turbulent liquid is investigated using Direct Numerical Simulation of turbulence (pseudo-spectral method) coupled with the Diffuse Interface Model to track the droplet interface. We explored a wide range of Weber numbers (ratio between inertial forces and surface tension) always limiting the analysis to cases of non-breaking droplets where the droplet and the surrounding fluid have the same density and viscosity. We quantify turbulence modulation across the interface in terms of velocity fluctuations and turbulent kinetic energy, showing that turbulence is always weaker inside the droplet. We also determine how the turbulent kinetic energy budget terms are influenced by the surface tension and how the local vorticity is affected by the presence of the interface.

3:46PM D23.00008 Turbulence close to the critical point of a fluid , GAUTIER VERHILLE, CECILE LACHIZE, PATRICE LE GAL, IRPHE - Aix-Marseille univ. - CNRS — Most of experiments in turbulence deal with liquid or gas. With classical fluids it is quite difficult to have both a high Reynolds number and a Mach number high enough to have compressible effects ($Ma \gtrsim 0.3$). In water the sound speed is too large to permit compressible effects ($c \sim 1500$m/s at room temperature and atmospheric pressure) and in air the viscosity is not so small ($\nu \sim 10^{-5}$m^2/s) so it is difficult to have high Reynolds number in a laboratory experiments. On the contrary, a fluid close to its critical point has a small kinematic viscosity, typically 20 times smaller than the water viscosity for SF6, and a small sound speed as the compressibility diverges, $c \sim 70$m/s for SF6. Other properties of the fluid are diverging close to the critical point, as the correlation length of the density fluctuation and other goes to zero, as the thermal conductivity. We present here the first study of the modification of a turbulent flow close to the critical point. This flow is created in a rotor stator cavity, a one disk version of the "french washing machine," in a pressurized and thermalized vessel filled up with SF6. Pressure and velocity measurements show an increase of the large scale dynamic whereas the inertial range does not seem to be affected.

3:59PM D23.00009 Chaos Synchronization in Navier-Stokes Turbulence[1] , CRISTIAN C. LALESCU, CHARLES MENEVEAU, GREGORY L. EYINK, The Johns Hopkins University — Chaos synchronization (CS) has been studied for some time now (Pecora & Carroll 1990), for systems with only a few degrees of freedom as well as for systems described by partial differential equations (Boccaletti et al. 2002). CS in general is said to be present in a pair of coupled dynamical systems when a specific property of each system has the same time evolution for both, even though the evolution itself is chaotic. There have been some studies of CS for systems with an infinite number of degrees of freedom (Kocarev et al. 1997), but CS for Navier-Stokes (NS) turbulence seems not to have been investigated so far. We focus on the synchronization of the small scales of a turbulent flow for which the time history of large scales is prescribed. We present DNS results which show that high-wavenumbers in turbulence are fully slaved to modes with wavenumbers up to a critical fraction of the Kolmogorov dissipation wavenumber. We compare our results with related ideas of "approximate inertial manifolds." The motivation for our work is to study deeply sub-Kolmogorov scales in fully developed turbulence (Schumacher 2007), which we show are recoverable even at very high Reynolds number from simulations that only resolve down to about the Kolmogorov scale.

[1]This work is supported by the National Science Foundation's CDI-II program, project CMMI-0941530.

Sunday, November 18, 2012 2:15PM - 4:25PM –

Session D24 Acoustics II: Jets and Cavities 30E - Chair: Meng Wang, University of Notre Dame

2:15PM D24.00001 Near and Far Field Acoustic Pressure Skewness in a Heated Supersonic Jet[1]

, EPHRAIM GUTMARK, PABLO MORA, JEFF KASTNER, NICK HEEB, University of Cincinnati, KAILAS KAILASANATH, JUNHUI LIU, Naval Research Laboratory, UNIVERSITY OF CINCINNATI COLLABORATION, NAVAL RESEARCH LABORATORY COLLABORATION — The dominant component of turbulent mixing noise in high speed jets is the Mach wave radiation generated by large turbulent structures in the shear layer The Over-All Sound Pressure Level (OASPL) in the far field peaks in a direction near the Mach wave angle. "Crackle" is another important component of high speed jet noise. Crackle cannot be recognized in the spectrum of the acoustic pressure signal, but it appears in the temporal waveform of the pressure as sharply rising peaks. Skewness levels of the pressure and dP/dt have been used as a measure of crackle in high specific thrust engines and rockets. In this paper, we focus on recognizing a technique that identifies the impact of different test conditions on the near-field and far-field statistics of the pressure and dP/dt signals of a supersonic jet with a design Mach number of Md=1.5 produced by a C-D conical nozzle. Cold and hot jets, T0=300K and 600K, are tested at over, design, and under-expanded conditions, with NPRs=2.5, 3.671, 4.5, respectively. Second, Third and Forth order statistics are examined in the near and far fields. Rms, skewness and kurtosis intensity levels and propagation are better identified in the dP/dt than in the pressure signal. Statistics of the dP/dt demonstrate to be a better measure for crackle.

[1]Project funded by ONR grant.

2:28PM D24.00002 The Sudden Onset of Acoustic Effects with Increasing Nozzle-Chevron Length In a Mach 0.9 Jet

, RYAN A. FONTAINE, Department of Mechanical Science & Engineering, University of Illinois, JOANNA M. AUSTIN, GREGORY S. ELLIOTT, Department of Aerospace Engineering, University of Illinois, JONATHAN B. FREUND, Departments of Mechanical Science & Engineering and Aerospace Engineering, University of Illinois — The addition of chevrons to the exit of a jet has been found to reduce far-field noise, making it an effective passive noise reduction technique. The selection of specific designs, however, is challenging since mechanics models provide insufficient guidance and computations remain far too expensive for trial-and-error parameter exploration. To assess the design optimization problem, we have experimentally evaluated parametric dependence of sound on chevron geometry. Specifically, rapid prototyping is used to change the model geometries and study the relationship between chevron length and the far-field jet noise. We show that the effect of the chevron starts relatively abruptly for lengths less than 4 mm, which is well less than the 16 mm length selected as a starting point from previous studies. Thus a small increase in the chevron length can produce a large change in the far-field sound. However, for longer chevrons the sound is insensitive to the length, which has important implications for design optimization. Additionally, other flow diagnostics are performed to better understand the flow characteristics resulting in the observed changes in jet noise.

2:41PM D24.00003 Far-field noise predictions of imperfectly expanded jet flows

, JUNHUI LIU, KAILAS KAILASANATH, Naval Research Laboratory, NICHOLAS HEEB, DAVE MUNDAY, EPHRAIM GUTMARK, University of Cincinnati — The far-field noise levels of imperfectly expanded jet flows are predicted using the Ffowcs Williams & Hawkings (FW-H) surface integral approach, where the information on the integral surfaces are generated from large-eddy simulations. Three FW-H surfaces are used to test the dependence of the far-field noise prediction on the location of the integral surfaces. The near-field pressure distributions on these FW-H surfaces are first examined to see if those surfaces are located in the acoustic propagation region. The variations of the monopole and dipole sources on the integral surfaces are also examined. The contribution of the shock-associated noise to the far-field noise level is well predicted. The difference between numerical predictions and measurements is within 1.0dB, whereas the contribution of the mixing noise is within 2.0dB. The contribution of the end cap is found to be small, but this contribution is sensitive to the mesh size used in the integration.

2:54PM D24.00004 Noise source detection and measurement in a supersonic air jet using Ultra-high Speed Rainbow Schlieren Deflectometry[1]

, MANIK RAJORA, University of Louisiana at Lafayette, AJAY AGRAWAL, WILLIAM MITCHELL, PANKAJ KOLHE, University of Alabama — Supersonic jets emit noise from various regions including the shear layer containing vortical structures, various shock cell structures in the near field and the downstream jet core breakdown region. Sound waves emitted from these various regions interact with each other and produce distinct noise spectra away from the jet, which depends upon the measurement location. Typically sound is detected by intrusive probes that provide measurements at a specific location, which makes it difficult to identify the origination point of such noise in a supersonic jet. In this study, an ultra-high speed Rainbow Schlieren Deflectometry (RSD) technique has been developed to optically visualize not only the supersonic jet flow but also the sound waves emanating from it in real time. Color schlieren images are acquired at up to 250,000 frames per second to capture the sound wave propagation with adequate spatial resolution. Optical components of the system were optimized to improve the spatial and temporal resolutions and hence, the schlieren video quality. To the best of our knowledge, this is the first time sound wave propagation from supersonic jets has been recorded in real time on a schlieren video. Acquired color schlieren images are amenable to quantitative analysis, and can provide data on sound power and sound wave frequency across the whole field.

[1]This project was funded by NSF REU 1062611 and Department of Energy for Institue for Sustainable Energy EE003134

3:07PM D24.00005 Unstructured Large Eddy Simulations of Hot Supersonic Jets from a Chevron Nozzle[1]

, GUILLAUME BRÈS, Cascade Technologies Inc., JOSEPH NICHOLS, SANJIVA LELE, Stanford University, FRANK HAM, Cascade Technologies Inc. — Large eddy simulations (LES) are performed for heated supersonic turbulent jets issued from a converging-diverging round nozzle with chevrons. The unsteady flow processes and shock/turbulence interactions are investigated with the unstructured compressible flow solver "Charles" developed at Cascade Technologies. In this study, the complex geometry of the nozzle and chevrons (12 counts, 6° penetration) are explicitly included in the computational domain using unstructured body-fitted mesh and adaptive grid refinement. Sound radiation from the jet is computed using an efficient frequency-domain implementation of the Ffowcs Williams–Hawkings equation. Noise predictions are compared to experimental measurements carried out at the United Technologies Research Center for the same nozzle and operating conditions. The initial blind comparisons show good agreement in terms of spectra shape and levels for both the near-field and far-field noise. The current results show that the simulations accurately capture the main flow and noise features, including the shock cells, broadband shock-associated noise and turbulent mixing noise. Additional analysis of the large database generated by the LES is ongoing, to further investigate jet noise sources and chevron effects.

[1]This work is supported by NAVAIR grant N68335-11-C-0026 managed by Dr. John Spyropoulos. The simulations were carried out at DoD supercomputer facilities in ERDC and AFRL as part of the HPC Challenge Project NAWCP30952C5.

3:20PM D24.00006 Acoustic response of heated jets to nozzle-upstream perturbations[1]

, MATTHIAS IHME, YEE CHEE SEE, GHOBAD AMINI, University of Michigan — Effects of nozzle-upstream entropy perturbations on the acoustic radiation from heated jets are investigated. For this, a model problem is considered, in which a gas-turbine combustor discharges reaction products through a converging nozzle into the ambient environment. The turbulent reacting flow field in the combustor is computed using large-eddy simulation, and the unsteady flow-field at the combustor exit is extracted to provide realistic inflow conditions to the jet-flow simulation. To study the indirect coupling process, arising from the interaction of the combustion-generated entropy fluctuations with the adverse pressure gradient through the nozzle, a linearized Euler formulation is employed. Parametric studies are performed to investigate effects of frequency and amplitude of the nozzle-upstream entropy perturbations on the jet instability and the jet noise directivity. It is shown that the excitation near the preferred shear-layer instability leads to strong acoustic radiation in the jet-forward direction, and the radiation angle decreases with decreasing excitation frequency.

[1]Financial support through the Office of Naval Research under Grant No. N00014-10-1-0717 is gratefully acknowledge.

3:33PM D24.00007 Active Control of Jet Noise Using Observable Inferred Decomposition and Large Window PIV , ZACHARY BERGER, MATTHEW BERRY, KERWIN LOW, Syracuse University, LAURENT CORDIER, BERND NOACK, University of Poitiers, SIVARAM GOGINENI, Spectral Energies, LLC., MARK GLAUSER, Syracuse University — In this investigation, we seek to find sources of noise created in the near-region of a highly subsonic jet, with a nozzle diameter of 2". Using large window PIV alongside simultaneous hydrodynamic and acoustic pressure, we focus on observing flow structures created in the collapse of the potential core. Correlations can be made between the low-dimensional velocity field (using POD) and the far-field acoustics in an effort to identify loud modes in the flow. An advanced reduced order model known as Observable Inferred Decomposition (OID) is used to form closed-loop controllers for noise reduction in the far-field. With this technique, we find low-dimensional representations of near-field velocity and far-field pressure – finding a linear mapping between the two fields. Then, we obtain acoustically optimized modes in the flow field and seek to drive these modes to zero using active control strategies. For flow control, synthetic jet actuators are used as shear layer excitation. A large range of tests are explored, varying Mach number and flow control configurations. Finally, large PIV windows will allow us to investigate several diameters of the flow field in the streamwise plane.

3:46PM D24.00008 Scaling model for nonlinear supersonic jet noise , WOUTIJN BAARS, CHARLES TINNEY, The University of Texas at Austin — Numerous endeavors have been undertaken to investigate nonlinear propagation of sound from jet flows in range-restricted environments. However, only weak observations of cumulative nonlinear effects have been made using these laboratory-scale setups, all the while being observed under full-scale conditions. The inconsistency is caused by the lack of rigor in understanding what the appropriate scaling parameters should be for producing measurable cumulative nonlinearities in laboratory-scale environments. A scaling model will be presented that one could use to guide future studies aimed at investigating this unique component of turbulent mixing noise. At first, the important length-scales for cumulative nonlinear waveform distortion – the shock formation distance and the acoustic absorption length – are written in terms of jet exit parameters. Their ratio, expressed as the effective Gol'dberg number, is a measure of the strength of nonlinear distortion relative to that of dissipation. By computing the individual length-scales and this dimensionless ratio for an experiment that is being designed, one can estimate the presence of cumulative nonlinear distortion beforehand.

3:59PM D24.00009 Large-eddy simulation of crackle in heated supersonic jets[1] , JOSEPH W. NICHOLS, SANJIVA K. LELE, Stanford University, Stanford, CA, FRANK E. HAM, Cascade Technologies, Inc., Palo Alto, CA, STEVE MARTENS, GE Global Research, Niskayuna, NY, JOHN T. SPYROPOULOS, Naval Air Systems Command, Patuxent River, MD — Crackle noise from heated supersonic jets is characterized by the presence of strong positive pressure impulses resulting in a strongly skewed far-field pressure signal (Ffowcs Williams et al., 1975). These strong positive pressure impulses are associated with N-shaped waveforms involving a shock-like compression, and thus is very annoying to observers when it occurs. In this talk, the origins of these N-shaped waveforms is investigated through high-fidelity large-eddy simulations (LES) applied to an over-expanded supersonic jet issuing from a faceted military-style nozzle. Two different levels of heating are considered. From the LES, we observe N-shaped waves associated with crackle to emerge directly from the jet turbulence. Furthermore, even at this extreme near-field location, we find that the emergent waves are already well-organized, having correlation over significant azimuthal distances.

[1]Computational resources were provided by a DoD HPCMP Challenge Project allocation at the ERDC and AFRL supercomputing centers.

4:12PM D24.00010 Predicting Acoustic Wave Generation and Amplification inside a Rectangular Cavity , RYAN SCHMIT, JAMES GROVE, Air Force Research Laboratory — Empirical and theoretical solutions to predict acoustic tones inside a rectangular cavity have been proposed throughout the years. A new theoretical understand is being developed that can now explain at the minimum the cavity tonal frequency response and possibly the self amplification of the acoustic tones. Current simulated results based on acoustic wave motion inside a cavity utilizing this theoretical understanding have produced results that compare well with experimental data all ready taken.

Sunday, November 18, 2012 2:15PM - 4:25PM –

Session D25 Flow Control: Aerodynamics 31A - Chair: Ari Glezer, Georgia Institute of Technology

2:15PM D25.00001 Rotorcraft Fuselage Flow Control Using Plasma Streamwise Vortex Generators[1] , DUSTIN COLEMAN, FLINT THOMAS, University of Notre Dame — Active flow control, in the form of dielectric barrier discharge (DBD) plasma actuators, is applied to a NASA ROBIN-mod7 generic rotorcraft fuselage model. The model is considered in what would be a typical cruise position *i.e.* a nose down position at $\alpha = -5^{\circ}$. This configuration gives rise to a massive 3-D flow separation over the aft ramp section of the fuselage, characterized by two counter-rotating, streamwise vortices. The control objective is to minimize these concentrated vortices by means of flush fuselage-mounted plasma streamwise vortex generators (PSVGs), and consequently, reduce the form drag of the vehicle. Experiments were conducted at freestream Mach and Reynolds numbers of $M_{\infty} = 0.12$ and $\mathrm{Re}_L = 2.65$ million, respectively. Aerodynamic loads under both natural and controlled conditions were acquired through use of an ATI Mini40 6-component force sensor. The pressure field on the ramp section was monitored by a 128 count static pressure array. Likewise, the flow field was captured by time-resolved PIV wake surveys. Results are compared with previous studies that utilized active flow control by way of pulsed jets or combustion actuators.

[1]This work is supported under NASA Cooperative Agreement NNX10AM32G

2:28PM D25.00002 Numerical simulations of a vertical tail of a commercial aircraft with active flow control[1] , MICHEL RASQUIN, Argonne National Laboratory and University of Colorado Boulder, JEFFREY MARTIN, KENNETH JANSEN, University of Colorado Boulder — A series of numerical simulations of a realistic vertical tail of a commercial aircraft, with a tapered swept stabilizer and a rudder, is considered in this work with application of flow control. Flow control is known to have the capacity to augment the streamwise momentum near the rudder suction peak where separation is typically observed to limit rudder effectiveness for high deflection angles. Specifically, we use Delayed Detached Eddy Simulations (DDES) to study the interaction of a cross flow with an array of 24 synthetic jets for a 0° angle of attack, a 30° deflection angle and a Reynolds number of 7×10^5. We concentrate our analysis on the influence of the spacing between successive active jets in the spanwise direction. Indeed, our current simulations suggest that doubling the number of active jets at a lower Reynolds number improves the lateral force while opposite effect is observed at the considered Reynolds number when using the same size jets. These simulations offer insight into the fundamental physics of the flow structures in the vicinity of the synthetic jets by accurately resolving the complete synthetic jet pathway and the vorticity plume where the jet structures interact with each other and with the primary flow.

[1]The Boeing Company and the Argonne Leadership Computing Facility are acknowledged for their support and resources through the INCITE program.

2:41PM D25.00003 Lift enhancement with extremum seeking control for a low-aspect-ratio wing , KUNIHIKO TAIRA, Florida State University, CLARENCE ROWLEY, Princeton University, TIM COLONIUS, California Institute of Technology — We introduce active flow control around a low-aspect-ratio flat-plate wing at post-stall angles of attack for lift enhancement. The separated flow is numerically examined with the immersed boundary projection method and the actuator is modeled as periodic momentum injection. It is emphasized that the present control technique does not suppress separation but benefits from the formation of the leading-edge vortices to take place closer to the suction side of the wing. This positions the low-pressure cores of the leading-edge vortices directly above the wing to provide added lift. For a certain range of actuation frequency, the wake is found to lock onto a periodic state that provides lift enhancement. To seek this high-lift state, we design a feedback controller with the extremum-seeking algorithm to determine the optimal actuation frequency. We discuss the design methodology and explain how the dynamics of the wake vortices is modified to yield added lift.

2:54PM D25.00004 Aerodynamic Control of a Pitching Airfoil using Distributed Active Bleed[1] , JOHN KEARNEY, ARI GLEZER, Georgia Institute of Technology — Aero-effected flight control using distributed active bleed driven by pressure differences across lifting surface and regulated by integrated louver actuators is investigated in wind tunnel experiments. The interaction between unsteady bleed and the cross flows alters the apparent aerodynamic shape of the lifting surface by regulating the accumulation and shedding of vorticity concentrations, and consequently the distributions of forces and moments. The present experiments are conducted using a 2-D dynamically-pitching VR-7 airfoil model from pre- to post-stall angles of attack. The effects of leading edge bleed at high angles of attack on the formation and evolution of the dynamic stall vorticity concentrations are investigated at high reduced frequencies ($k > 0.1$) using PIV phase-locked to the airfoil's motion. The time-dependent bleed enables broad-range variation in lift and pitching moment with significant extension of the stall margin. In particular, bleed actuation reduces the extent of "negative damping" or pitching moment instability with minimal lift penalty.

[1]Supported by NTRC-VLRCOE, monitored by Dr. Mike Rutkowski

3:07PM D25.00005 Transitory Control of Separation over a Pitching Airfoil using Spatially-Compact Pulsed Actuation[1] , GEORGE T.K. WOO, ARI GLEZER, Georgia Institute of Technology — The dynamics of controlled three-dimensional attachment of transitory stall over a pitching 2-D airfoil is investigated in wind tunnel experiments using spanwise-compact (0.12S) pulsed actuation on time scales that are an order of magnitude shorter than the characteristic convective time scale. Actuation is effected using a surface-integrated pulsed, combustion-based fluidic actuators. The flow field above the airfoil and in its near wake is computed from multiple high-resolution PIV images in multiple cross stream planes that are obtained phase-locked to the actuation, and allow for tracking of vorticity concentrations. Transitory attachment spreads towards the outboard, unactuated flow domains and far exceeds the width of the actuator. The actuation results in the formation of 3-D vortical structures that are advected and shed into the near wake and induce spanwise variations in the sectional circulation. It is shown that coupling of the pulsed actuation to the airfoil's motion enhances the control authority of 3-D unsteady separation, can significantly mitigate the effects of dynamic stall and improve the unsteady aerodynamic lift and pitching moment.

[1]Supported by AFOSR.

3:20PM D25.00006 Numerical Study of Three Dimensional Mechanisms in Impulse Actuated Stall Control[1] , SIGFRIED HAERING, ROBERT MOSER, Unviersity of Texas at Austin — Dynamic stall control by means of anharmonic pulsed jet actuation may yield a simple means to significantly increase the performance of lifting surfaces and expand their flight envelope. Previous numerical studies of spanwise-uniform actuation regions have provided a highly detailed understanding of boundary layer reattachment mechanisms. Naturally, an equivalently stringent examination of three-dimensional effects of actuation is necessary to fully utilize this approach. To understand the spanwise spread of the actuation effect and the vortex dynamics of the boundary region at the edge of the actuated zone, a single spanwise-finite actuator is simulated using delayed detached eddy simulation (DDES). The results of these simulations may be used to design optimal actuator spacing, orientation, and scheduling.

[1]AFOSR

3:33PM D25.00007 Feedback control of a pitching and plunging airfoil via direct numerical simulation[1] , SCOTT DAWSON, STEVEN BRUNTON, CLARENCE ROWLEY, Princeton University — Feedback control is implemented in direct numerical simulations at a Reynolds number of 100 to allow a two-dimensional flat plate airfoil to track desired lift profiles using pitching and plunging motions. Robust controllers are designed using both classical models (Theodorsen) and empirical reduced-order models identified from direct numerical simulations. We investigate the capabilities of a variety of controllers for plunging motion and for pitching about different pitch axis locations. Effective control is achieved across a wide range of angles of attack, despite strongly nonlinear flow physics. The forces caused by rapid airfoil motion may be utilized to achieve high lift coefficients for short periods of time. It is also possible to track periodic lift profiles with average lift coefficients that are significantly greater than those achieved by a steady airfoil. The enhanced lift that arises at certain frequencies appears to be caused by favorable interaction of wake vortices. The ability of the controllers to reject gust disturbances and attenuate sensor noise is also investigated, which is relevant for the implementation of such controllers in an experimental setting.

[1]This work is supported by AFOSR grant FA9550-12-1-0075

3:46PM D25.00008 The Transient Aerodynamic Forces Effected by Trailing Edge Active Flow Control , DAN BRZOZOWSKI, The Boeing Company, JOHN CULP, ARI GLEZER, Georgia Institute of Technology — The transient aerodynamic forces effected by trailing edge flow control are investigated in wind tunnel experiments using a 2-DOF traverse which enables application of time-dependent external torque and forces by servo motors. The global aerodynamic forces and moments are regulated by controlling vorticity generation and accumulation near the surface using hybrid synthetic jet actuators. The time-histories of surface pressure and aerodynamic lift and pitching moment immediately following the application of flow control are measured using simultaneous pressure, force and velocity measurements that are taken phase-locked to the commanded actuation waveform. Circulation time history that is estimated from a PIV wake survey shows that the entire flow over the airfoil readjusts within about $1.5T_{\mathrm{CONV}}$, which is about two orders of magnitude shorter than the characteristic time associated with the controlled maneuver of the wind tunnel model. This illustrates that flow-control actuation can be typically effected on time scales that are commensurate with the flow's convective time scale, and that the maneuver response is primarily limited by the inertia of the platform.

3:59PM D25.00009 Parametric Investigation of Nanosecond Pulse Driven Dielectric Barrier Discharge Plasma Actuators for Aerodynamic Flow Control

, ROBERT DAWSON, JESSE LITTLE, University of Arizona — Nanosecond pulse driven dielectric barrier discharge plasma actuators are studied experimentally in quiescent atmosphere. Per unit length peak energy and instantaneous peak power are calculated using simultaneous voltage and current measurements. Electrical characteristics are evaluated as a function of peak voltage, pulse frequency, discharge length and dielectric thickness. Schlieren imaging of compression waves is used to provide a relative measure of discharge energy that is coupled to the near surface gas as heat for the same parameters. Characteristics of the DBD load have a substantial effect on voltage and current traces which are reflected in the peak energy and peak power. Both peak energy and compression wave strength depend primarily on dielectric thickness and secondarily on actuator length although this is not universal in the case of energy necessitating examination of alternative calculation strategies. Peak power is mainly dependent on actuator length which is inconsistent with schlieren data as expected. Higher pulse frequency produces higher pulse energy, but is primarily attributed to heating of the actuator and power supply components. This effect is mainly observed for short actuators. Pulse energy increases as peak voltage to the power 3.5. This behavior is similar to observations of energy and thrust for ac-DBD plasma actuators suggesting that aspects of lumped-element circuit models may be applicable for optimizing ns-DBD performance.

4:12PM D25.00010 On least-order flow decompositions for aerodynamics and aeroacoustics[1]

, MICHAEL SCHLEGEL, Berlin Institute of Technology, Germany, BERND R. NOACK, PETER JORDAN, Institute PPRIME, France — A generalisation of proper orthogonal decomposition (POD) for optimal flow resolution of linearly related observables is presented, as proposed in the identically named publication of Schlegel, Noack, Jordan, Dillmann, Groeschel, Schroeder, Wei, Freund, Lehmann and Tadmor (Journal of Fluid Mechanics 2012, vol. 697, pp. 367–398). This Galerkin expansion, termed "observable inferred decomposition" (OID), addresses a need in aerodynamic and aeroacoustic applications by identifying the modes contributing most to these observables. Thus, OID constitutes a building block for physical understanding, least-biased conditional sampling, state estimation and control design. From a continuum of OID versions, two variants are tailored for purposes of observer and control design, respectively. Three aerodynamic and aeroacoustic observables are studied: (1) lift and drag fluctuation of a two-dimensional cylinder wake flow, (2) aeroacoustic density fluctuations measured by a sensor array and emitted from a two-dimensional compressible mixing layer, and (3) aeroacoustic pressure monitored by a sensor array and emitted from a three-dimensional compressible jet. The most "drag-related," "lift-related" and "loud" structures are distilled and interpreted in terms of known physical processes.

[1]This work was partially funded by the DFG under grants SCHL 586/2-1 and ANR, Chair of Excellence, TUCOROM.

Sunday, November 18, 2012 2:15PM - 4:25PM –

Session D26 Reactive Flows II: Turbulent Combustion Modelling

31B - Chair: Tarek Echekki, North Carolina State University

2:15PM D26.00001 Numerical Investigation of a Piloted Premixed Jet Burner

, YUNTAO CHEN[1], MATTHIAS IHME[2], The University of Michigan at Ann Arbor — Premixed and partially premixed combustion technologies have the potential of reducing pollutant emissions and enhancing the overall combustion efficiency. However, the successful implementation of these technologies in practical systems requires a thorough understanding of the underlying combustion-physical phenomena. To predict partially premixed combustion, a three-stream progress-variable model has been developed. This model is applied to large-eddy simulations of the piloted premixed jet burner experiment of Dunn et al., and configurations PM1-{50,100,150} are considered. Comparisons with measurements for all three configurations are presented, and it is shown that this model is capable of capturing the characteristics of the flow-field, temperature, and compositional field.

[1]Graduate Student Research Assistant
[2]Assistant Professor

2:28PM D26.00002 Verification and Validation of a Chemical Reaction Solver Coupled to the Piecewise Parabolic Method[1]

, NITESH ATTAL, PRAVEEN RAMAPRABHU, University of North Carolina at Charlotte, JAHED HOSSAIN, University of Central Florida, VARAD KARKHANIS, University of North Carolina at Charlotte, SUKESH ROY, Spectral Energies LLC, JAMES GORD, Air Force Research Laboratory, MESBAH UDDIN, University of North Carolina at Charlotte — We present a detailed chemical kinetics reaction solver coupled to the Piecewise Parabolic Method (PPM [1]) embedded in the widely used astrophysical FLASH [2] code. The FLASH code solves the compressible Euler equations with a directionally split, PPM with Adaptive Mesh Refinement (AMR). The reaction network is solved using a library of coupled ODE solvers, specialized for handling stiff systems of equations. Finally, the diffusion of heat, mass, and momentum is handled either through an update of the fluxes of each quantity, or by directly solving a diffusion equation for each. The resulting product is capable of handling a variety of physics such as gas-phase chemical kinetics, diffusive transport of mass, momentum, and heat, shocks, sharp interfaces, multi-species mixtures, and thermal radiation. We will present results from verification and validation of the above capabilities through comparison with analytical solutions, and published numerical and experimental data. Our validation cases include advection of reacting fronts in 1-D and 2D, laminar premixed flames in a Bunsen burner configuration, and shock-driven combustion.
[1] P. Colella and P. R. Woodward, J. Comput. Phys. 54, 174-201 (1984). [2] B. Fryxell et al., Astrophys. J., Suppl. Ser. 131, 273 (2000).

[1]We acknowledge funding from Spectral Energies LLC.

2:41PM D26.00003 Nonlinear Principal Component Analysis for Combustion Large-Eddy Simulation

, HESSAM MIRGOLBABAEI, North Carolina State University, TAREK ECHEKKI, Department of Mechanical and Aerspace Engineering, North Carolina State University — Moment-based methods have been widely used in turbulent combustion modeling, by transporting a set of moments and reconstructing thermo-chemical scalars' statistics. Instead of ad hoc strategies to select these moments, more optimal moments have been proposed recently using principal component analysis (PCA). However, it is not evident that the linear PCA alone can represent the non-linear nature of the space representing thermo-chemical scalars' statistics effectively. As a nonlinear alternative to the classical PCA in which the linear transformation of the original space is constructed, kernel PCA(KPCA) is adapted in the present work, in that the original space is mapped into a Feature space where the intrinsic dimensionality can be linearly extracted. Re parameterization is performed based on one-dimensional turbulence (ODT) for the implementation of KPCA, for implementation in a novel large-eddy simulation (LES) approach, as well. This involves the solution of LES on a coarse-grid and fine-grained ODT solutions embedded in the LES domain. Parameters from the KPCA analysis can be used to evaluate key terms in the transport equation for the moments. These terms are tabulated using artificial neural networks (ANN).

2:54PM D26.00004 A Novel Principal Component Analysis-Based Acceleration Scheme for LES-ODT: An A Priori Study , TAREK ECHEKKI, HESSAN MIRGOLBABAEI, North Carolina State University — A parameterization of the composition space based on principal component analysis (PCA) is proposed to represent the transport equations with the one-dimensional turbulence (ODT) solutions of a hybrid large-eddy simulation (LES) and ODT scheme. An a priori validation of the proposed approach is implemented based on stand-alone ODT solutions of the Sandia Flame F flame, which is characterized by different regimes of combustion starting with pilot stabilization, to extinction and reignition and self-stabilized combustion. The PCA analysis is carried out with a full set of the thermo-chemical scalars' vector as well as a subset of this vector. The subset is made up primarily of major species and temperature. The results show that the different regimes are reproduced using only three principal components for the thermo-chemical scalars based on the full and a subset of the thermo-chemical scalars' vector. Reproduction of the source term of the principal component represents a greater challenge. It is found that using the subset of the thermo-chemical scalars' vector both minor species and the first three principal components source terms are reasonably well predicted.

3:07PM D26.00005 LES-ODT Simulations of Turbulent Reacting Shear Layers , ANDREAS HOFFIE, TAREK ECHEKKI, North Carolina State University — Large-eddy simulations (LES) combined with the one-dimensional turbulence (ODT) simulations of a spatially developing turbulent reacting shear layer with heat release and high Reynolds numbers were conducted and compared to results from direct numerical simulations (DNS) of the same configuration. The LES-ODT approach is based on LES solutions for momentum on a coarse grid and solutions for momentum and reactive scalars on a fine ODT grid, which is embedded in the LES computational domain. The shear layer is simulated with a single-step, second-order reaction with an Arrhenius reaction rate. The transport equations are solved using a low Mach number approximation. The LES-ODT simulations yield reasonably accurate predictions of turbulence and passive/reactive scalars' statistics compared to DNS results.

3:20PM D26.00006 Novel Strategies for Coupling 3D LES with ODT Solutions Based on Wavelet and Assimilation Methods , YUQIANG FU, TAREK ECHEKKI, North Carolina State University — The LES-ODT approach is a multi-scale framework for the simulation of turbulent reacting flows. It is based on the implementation of two solutions: 1) LES for continuity and momentum and 2) ODT for momentum and reactive scalars. The coupling is based on upscaling, which filters the ODT solution for the filtered density onto LES and upscaling, which corrects the ODT momentum solution to make it consistent with the LES solution. Wavelet-based downscaling uses the compound wavelet matrix method to substitute large-scale physics from LES onto the ODT solution, while maintaining the ODT residual sub-grid scale contribution intact. A Kalman filter based upscaling is used obtain a smooth filtered density field from ODT. The present study demonstrates the use of the upscaling and downscaling methods for a LES-ODT simulation of a reacting flow.

3:33PM D26.00007 FDF Simulation of a Realistic Gas Turbine Combustor , NASEEM ANSARI, University of Pittsburgh, ANSYS Inc., PATRICK PISCIUNERI, University of Pittsburgh, PETER STRAKEY, National Energy Technology Laboratory, PEYMAN GIVI, University of Pittsburgh — The unstructured scalar filtered mass density function (SFMDF) is employed for the large eddy simulation (LES) of a realistic swirl flame combustor. This is the PRECCINSTA experimental burner from the German Aerospace Center (DLR) [1]. This burner is a reasonable representation of an industrial gas turbine combustor and has been the subject of broad experimental and computational investigations. To keep the geometrical complexity of the burner intact, the fuel injection holes are meshed along with radial swirler vanes and the mixing zone prior to the nozzle exit. Combustion chemistry is modeled with a reduced mechanism featuring 16 species and 12 reaction steps. The simulated data are analyzed by comparison of the Reynolds-averaged statistics with experimental data and show excellent agreement. This demonstrates the capability of FDF for LES of complex flows, and warrants future applications of the methodology for LES of practical combustor configurations.

[1] W. Meier, P. Weigand, X.R. Duan, and R. Giezendanner-Thoben. Detailed characterization of the dynamics of thermoacoustic pulsations in a lean premixed swirl burner. *Combust. Flame*, **150**:2-26, (2007).

3:46PM D26.00008 Petascale FDF Large Eddy Simulation of Reacting Flows , PATRICK PISCIUNERI, Mechanical Engineering and Materials Science, University of Pittsburgh, S. LEVENT YILMAZ, Center for Simulation and Modeling, University of Pittsburgh, PEYMAN GIVI, Mechanical Engineering and Materials Science, University of Pittsburgh — A novel computational methodology, termed "Irregularly Portioned Lagrangian Monte Carlo-Finite Difference" (IPLMCFD) is developed for large eddy simulation (LES) of turbulent flows. This methodology is intended for use in the filtered density function (FDF) formulation and is particularly suitable for simulation of chemically reacting flows on massively parallel platforms. The IPLMCFD allows for tremendous improvements in scalability, and is the key enabler of petascale computations. The methodology is employed for LES of several flame configurations.

3:59PM D26.00009 Filtered Density Function for Large Eddy Simulation of Local Entropy Generation in Turbulent Reacting Flows , MEHDI SAFARI, M. REZA H. SHEIKHI, Northeastern University — Analysis of local entropy generation is an effective means to investigate sources of efficiency loss in turbulent combustion from the standpoint of the second law of thermodynamics. A methodology, termed the "entropy filtered density function" (En-FDF), is developed for large eddy simulation (LES) of turbulent reacting flows to include the transport of entropy. The filtered form of entropy transport equation contains several unclosed entropy generation terms that contribute to efficiency losses in turbulent combustion. The closure is provided by the En-FDF, which embodies the complete statistical information about entropy variations within the subgrid scale. An exact transport equation is developed for the En-FDF. The unclosed terms in this equation are modeled by considering a system of stochastic differential equations. The modeled En-FDF transport equation is solved by a Lagrangian Monte Carlo method. The En-FDF is applied to a turbulent shear layer and validated by comparing with results obtained from direct numerical simulation of the same layer. The methodology is also employed to study local entropy generation in turbulent flames. The results are compared with the experimental data.

4:12PM D26.00010 LES-CMC Approach for Dilute Reacting Sprays , SANTANU DE, BOXIONG CHEN, SEUNG HYUN KIM, Michigan Technological University — Large eddy simulation (LES) of dilute reacting spray jets is carried out using the conditional moment closure (CMC) method. Over the past few decades, LES-CMC has shown its capabilities in describing turbulence-chemistry interactions in gas-phase non-premixed reactive flows. In this study, we address some modeling complexities arising due to the presence of multiple phases. Most of the previous CMC studies have either neglected the effects of evaporation on conditional moments or used simplified models. Only recently, fundamental equations for the CMC modeling of spray combustion have been derived. The resulting CMC equations contain several correlation terms associated with phase changes. Here, an attempt is made to model those unclosed terms. A stochastic method is used to generate the SGS fluctuations in gas-phase reactive scalars consistent with the framework of a mixture-fraction-based combustion model. The SGS fluctuations of the gas-phase temperature and composition, seen by droplets, are used to refine the estimates of the interphase heat/mass transfer rates. Results from the LES-CMC approach for both the gas- and liquid phases are extensively validated against available experimental measurements.

Sunday, November 18, 2012 2:15PM - 4:25PM –

Session D27 Supersonic Flows I 31C - Chair: Xiaolin Zhong, University of California, Los Angeles

2:15PM D27.00001 Characterization of the Shear Layer in a Mach 3 Shock/Turbulent Boundary Layer Interaction , CLARA HELM, University of Maryland, College Park, STEPHAN PRIEBE, Princeton University, PIERRE DUPONT, Groupe Supersonique, IUSTI, UMR CNRS 6595 and Aix-Marseille Univ., Marseille, France, PINO MARTIN, University of Maryland,College Park — A detailed characterization of the shear layer in a direct numerical simulation of a Mach 3 shock/turbulent boundary layer interaction for a 24° ramp is presented. The behaviour of the shear layer as a plane mixing layer will be demonstrated through similarity profiles of the mean velocity and Reynolds stresses. The existence of large scale coherent eddies associated with the Kelvin-Helmholtz structures characteristic of plane mixing layers is investigated. An estimation of the time and length scales associated with these eddies is conducted. Also presented is evidence of the modulation of the fluctuating intensities in the shear layer by the low-frequency motion of the shock. This work is supported by the Air Force Office of Scientific Research under grant AF/9550-10-1-0164.

2:28PM D27.00002 Shock Boundary Layer Interaction Sensitivity to Upstream Geometric Perturbations , LAURA CAMPO, DAVID HELMER, JOHN EATON, Stanford University — Shock boundary layer interactions (SBLIs) can have drastic effects on the performance of external aerodynamics and propulsion systems in high speed flight vehicles. In such systems, the upstream and boundary conditions of the flow are uncertain, and the sensitivity of SBLIs to perturbations in these conditions is unknown. The sensitivity of two SBLIs – a compression corner interaction and an incident shock interaction – to small geometric perturbations was investigated using particle image velocity measurements. Tests were performed in a continuously operated, low aspect ratio, Mach 2.1 wind tunnel. The shock was generated by a 1.1mm high 20° wall-mounted compression wedge, and various configurations of small ($h < 0.2\delta$) steady bumps were introduced upstream on the opposite wall. 100 perturbed cases were tested in order to generate a dataset which is well suited for validation of CFD codes. Both SBLIs were very sensitive to perturbations in a given region and insensitive to perturbations outside of it. Depending on the location of the perturbations, the compression corner interaction could be significantly strengthened or weakened. The position of the incident SBLI was also a strong function of both the location and size of the upstream perturbations.

2:41PM D27.00003 LES Study of Shock Wave and Turbulent Boundary Layer Interaction[1] , JUSTINE LI, STEPHAN PRIEBE, PINO MARTIN, University of Maryland, College Park — The large eddy simulation (LES) of a 24° compression ramp shock wave and turbulent boundary layer interaction (STBLI) is presented. This work builds on previous work on the direct numerical simulation (DNS) of STBLI with similar incoming boundary layer flow conditions (Priebe and Martín, JFM 2012). The fully-turbulent inflow boundary layer is at Mach 2.9 and the Reynolds number based on momentum thickness is $\mathrm{Re}_\theta = 2900$. The LES data cover a sufficiently long time to statistically resolve the low-frequency aperiodic cycle characteristic of supersonic STBLI. We present the characterization of the dynamics in the downstream separated flow.

[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.

2:54PM D27.00004 Large-Eddy Simulation of a Shock Train in a Duct with Side Walls , BRANDON MORGAN, KARTHIK DURAISAMY, SANJIVA LELE, Stanford University — Large-eddy simulation (LES) is utilized to investigate the three-dimensionality of a shock train in a constant-area isolator model with fully resolved side walls ($M_\infty = 1.61$, $\mathrm{Re}_\theta \approx 1660$). Flow conditions and geometry are selected to match experimental conditions investigated by Carroll (1988); although Reynolds number is reduced to ensure adequate mesh resolution. Simulations with spanwise periodic boundary conditions are first performed, the results of which are compared to experiment and validated with a three-level grid refinement study. The same shock train interaction is then simulated in a three-dimensional, low-aspect ratio rectangular duct geometry with particular emphasis placed on characterizing secondary corner flows and the effects of these corner flows on the location and structure of the shock train. It is found, for instance, that location of the initial shock is particularly sensitive to the effects of spanwise confinement. Most significantly, it is observed that the same pressure ratio which results in a stable shock train with periodic boundary conditions may result in isolator unstart when side-wall effects are fully resolved.

3:07PM D27.00005 Toluene PLIF temperature and pressure imaging in supersonic flows[1] , VICTOR MILLER, Stanford University, MIRKO GAMBA, University of Michigan, M. GODFREY MUNGAL, University of Santa Clara, Stanford University, RONALD K. HANSON, Stanford University — Planar laser-induced fluorescence (PLIF) of toluene is used to image temperature, T, and pressure, P, in supersonic flows. Toluene fluorescence is broadband and exhibits a red-shift at elevated temperatures, enabling two-camera, dual-band imaging of temperature. With the temperature field known, the original LIF signal from a single camera is converted to pressure. This technique is demonstrated by imaging Mach 2.3 flow of nitrogen seeded with 0.5% toluene (by volume) over a wedge and a cylinder; PLIF-imaged T and P fields are compared to computed solutions of these flowfields, and preliminary comparisons show promise.

[1]Work is supported by Department of Energy and its Predictive Science Academic Alliance Program

3:20PM D27.00006 A Multi-Zone and Cut-Cell Method for High-Order Numerical Simulations of Compressible Flow Over Arbitrary Geometries , PATRICK GREENE, JEFF ELDREDGE, XIAOLIN ZHONG, JOHN KIM, University of California, Los Angeles — The primary goal of our work is to study the effects of isolated roughness elements on the stability of hypersonic flow. To achieve this goal, we have been developing a high-order finite-difference compressible Navier-Stokes solver with the ability to simulate high-speed flow over arbitrary roughness geometries. A high-order cut-cell method is used to impose the roughness geometry on a simple Cartesian grid. The main idea behind the cut-cell method is to switch to a high-order non-uniform one-sided finite-difference stencil whenever the normally used stencil will cross a fluid-solid interface. In addition, a multi-zone method is implemented to provide more precise control over the placement of grid points compared to a single grid with clustering. The method allows a coarse grid to be placed over the entire domain with smaller high-resolution grids placed in regions where complex flow physics are expected. Results are presented for hypersonic flow over a cylindrical roughness on a flat plate. The roughness height is on the same order of magnitude as the boundary layer thickness. The results will be compared to recent experiments.

3:33PM D27.00007 Rainbow Schlieren Deflectometry Measurements With a Pulse Detonation Engine , CHARLES DESIO, The University of Alabama, CHRISTOPHER STEVENS, Innovative Scientific Solutions Inc., RUDY JOHNSON, Air Force Research Lab, Wright Patterson Air Force Base, SEMIH OLCMEN, The University of Alabama — Unsteady, under-expanded, Mach five flow is observed and measured with the use of a rainbow schlieren deflectometry technique. Measurements are taken on a pulse detonation engine system at the exit of a 0.5 inch diameter pre-detonation tube and at the exit of a two inch diameter thrust tube. The predetonator is fueled with propane/nitrous oxide, and the thrust tube is fueled with hydrogen/air. Collimated light illuminates the flow field after passing a 100 micron slit in a conventional schlieren setup. The collected light is focused on a color filter, and light ray deflections fall on different color bands. The defections can be calculated using a simple calibration. Angular deflections due to density changes in the flow are recorded by a high-speed, color camera. Density gradients are calculated along the flow axis as well as perpendicular to the flow axis. Structures observed without post-processing include: Shock waves, flame fronts, Mach disk, and shock diamonds. The color schlieren system is simple to set up and use, relatively inexpensive to purchase, and produces accurate, nonintrusive flow data.

3:46PM D27.00008 Planar Imaging of Mach 3 Hypermixer Flowfields with Varying Geometry[1]

, ROSS BURNS, NOEL CLEMENS, The University of Texas at Austin — At the high Mach number associated with hypersonic flight, potentially excessive pressure loads and changes in air chemistry necessitate supersonic flow within a scramjet combustor. A form of mixing enhancement is therefore required to enable proper mixing of the fuel and air streams and maintain efficient combustion. Hypermixers have shown promise as an effective mixing enhancement strategy, utilizing streamwise vorticity to enhance large scale transport and micromixing rather than relying solely on turbulence. An experimental investigation of several strut-based Mach 3 hypermixing flowfields is being conducted, concentrating on the effect of geometric variations (ramp angle and spacing) on the flowfield mixing characteristics. Global flow features are examined through the use of planar laser scattering (PLS) and two-component particle image velocimetry (PIV). The evolution of streamwise vortical structures is observed at different streamwise locations using stereoscopic PIV. Finally, the interaction of these vorticies with an injected scalar is studied by combining the use of two- and three-component PIV with planar laser-induced fluorescence (PLIF).

[1]This work was supported by NASA Fundamental Aeronautics Program.

3:59PM D27.00009 Numerical investigation of non-equilibrium effects in hypersonic turbulent boundary layers

, PILBUM KIM, JOHN KIM, XIAOLIN ZHONG, JEFF ELDREDGE, University of California, Los Angeles — Direct numerical simulations of a spatially developing non-equilibrium hypersonic turbulent boundary layer have been conducted. A pure oxygen flow over a non-catalytic flat plate at the wall temperature, $T_w = 1,000\ K$, is considered. The boundary edge conditions are given as $M_e = 9.1$, $T_e = 792\ K$, and $P_e = 6,565\ Pa$, which are corresponding to flow conditions around a blunt wedge with a $3.174\ mm$ radius and 7° half angle at $M_\infty = 15.3$, $T_\infty = 285\ K$, and $P_\infty = 664\ Pa$. The initial conditions are obtained from a turbulent boundary layer simulation of a perfect gas. In addition, the species concentrations and vibrational temperature at the inlet are prescribed as equilibrium concentrations and the translational/rotational temperature, respectively. The data samples are collected at a downstream location at which a statistically stationary state has been achieved. From the collected data set, turbulence quantities are computed and compared with those from a perfect gas simulation in order to investigate the effects of thermal and chemical non-equilibrium on turbulent boundary layers. Those comparisons will be reported.

4:12PM D27.00010 Transition within a hypervelocity boundary layer on a 5-degree half-angle cone in freestream air/CO_2 mixtures

, JOSEPH JEWELL, California Institute of Technology, ROSS WAGNILD, Sandia National Laboratories, IVETT LEYVA, Air Force Research Laboratory, GRAHAM CANDLER, University of Minnesota, JOSEPH SHEPHERD, California Institute of Technology — The effect of freestream CO_2 content on transition in hypervelocity flow over a slender cone was investigated in experiments and computations. Experimental data were obtained in Caltech's T5 reflected shock tunnel. The model was a 5 degree half-angle sharp cone instrumented with thermocouples, providing heat transfer measurements from which transition locations were determined by comparison with laminar and turbulent heat flux correlations. Four carbon dioxide/air gas mixtures were tested at reservoir enthalpies from 7–10 MJ/kg and reservoir pressures from 55–60 MPa to attempt to reproduce the largest shift in transition location implied by computations using the semi-empirical e^N approach. By mass fraction of carbon dioxide these mixtures were 0.0 (e.g. all air), 0.5, 0.75, and 1.0. For tests at an enthalpy of ∼9.2 MJ/kg, transition delays of up to 30% in terms of x-location, 38% in terms of edge Reynolds number, and 140% in terms of the Reynolds number evaluated at reference conditions were documented for increasing CO_2 mass fractions compared with similar experiments in air.

Sunday, November 18, 2012 2:15PM - 4:25PM –

Session D28 Free-Surface Flows II 32A - Chair: Javier Rodriguez-Rodriguez, Carlos III University of Madrid

2:15PM D28.00001 Numerical simulations of transient air entrainment by rough and smooth plunging jets

, KEN KIGER, University of Maryland, NABIL KHAROUA, LYES KHEZZAR, The Petroleum Institute — Plunging jets are intimately linked to the process of air or gas entrainment into liquid pools, and can play either a beneficial or detrimental role in many environmental and industrial flows. The purpose of the present work is to assess the capability of combined LES/VOF algorithms to simulate water/air plunging jet flows, starting with the transient impact of the free jet, initial cavity formation, pinch off, and evolution towards a continuous entrainment phase. We focus on what happens in the transient impact phase for weakly and highly disturbed jets, operating with impact conditions of $Re = UD/\nu = 10,500$, $We = \rho U^2 D/\sigma = 300$ and $Fr = U^2/gD = 83$. In particular, the study investigates the ability of the simulations to capture liquid surface instabilities and the influence of the exiting jet turbulence content on the entrainment behavior. The results indicate that the qualitative behavior of the entrainment process follows very closely what is observed in experiments, with the rough jet exhibiting surface instabilities at impact that are not present in the smooth jet. These have an effect on the development of the initial air cavity and interfacial area, leading to a doubling of the interfacial area for a nominally similar entrained volume of air.

2:28PM D28.00002 Lagrangian observations of acceleration and bubble dynamics in plunging breakers

, MIGUEL CANALS, ANDRE AMADOR, University of Puerto Rico at Mayaguez — Understanding the three-dimensional structure of plunging waves is one of the most difficult problems in fundamental fluid dynamics. In this presentation we provide an analysis of field data collected in breaking waves using novel Lagrangian drifters with a diameter of 5-10 cm and equipped with miniature HD cameras and inertial measurement units. These drifters were deployed, using a personal watercraft, into the breaking region of waves ranging from 1-5 meters in height. We analyze in detail the time series of particle acceleration and rotation and how these quantities relate to the imagery captured by the camera aboard the drifters. This data represents the first dedicated study of the three-dimensional particle dynamics of plunging breakers. Going beyond the basic statistical analysis of the acceleration data, we make an attempt at characterizing the intensity of the wave breaking process using the bubble size and characteristics obtained from the HD video images. We also attempt to relate the spectral statistics of acceleration and particle rotation to existing Lagrangian turbulence models in the hopes of obtaining estimates of the kinetic energy dissipation in breaking waves, while taking into account the unsteady and heterogeneous nature of the turbulent flow.

2:41PM D28.00003 An Experimental Study of Droplets Produced by Plunging Breakers[1]

, D. WANG, University of Maryland, D. DAI, First Institute of Oceanography, China, X. LIU, J.H. DUNCAN, University of Maryland — The production of droplets by breaking water waves greatly affects the heat, mass and momentum transfer between the atmosphere and the sea surface. In this study, the production of droplets by mechanically generated breaking water waves was explored in a wave tank. The breakers were generated from dispersively focused wave packets (average frequency 1.15 Hz) using a programmable wave maker. Two overall wave maker amplitudes were used to create a strong spilling and a strong plunging breaker. The profile histories of the breaking wave crests along the center plane of the tank were measured with a cinematic laser-induced fluorescence technique, while the droplet diameter distributions and motions were measured at different locations along a horizontal line, which is 1 cm above the maximum height of the wave crest, using a double-pulsed cinematic shadowgraph technique. It is found that droplets are primarily generated when the plunging jet of the wave generates strong turbulence during impact with the wave's front face and when large air bubbles, entrapped during the plunging process, rise to the free surface and pop. The differences between the generation mechanisms in spilling and plunging breakers is highlighted.

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

2:54PM D28.00004 Quantification of air entrainment in a turbulent breaker from Bubble Image Velocimetry (BIV) data , JAVIER RODRIGUEZ-RODRIGUEZ, Carlos III University of Madrid, Spain, JUAN LASHERAS, University of California, San Diego — One of the most difficult to quantify features of free surface turbulent flows is the amount of air that they engulf. Among other reasons, the large void fractions found in these flows preclude the application of many diagnostic techniques commonly used in fluid mechanics. In this work, we investigate the possibility of taking advantage of the large void fraction found in a turbulent breaker to simultaneously characterize the velocity field and the flux of air entrained across the mean free surface. To that end, we use a technique known as Bubble Image Velocimetry (BIV) that consists of applying standard PIV algorithms to a sequence of images of a bubble laden flow. Under certain conditions, that will be explained in this talk, the bubbles follow the motion of the large turbulent structures that are the main responsible for the entrainment of air bubbles. Thus tracking the motion of the bubbles, and measuring their number and volume distribution, it is possible to evaluate the flux of air across the mean free surface. The results obtained with this method are compared with those found in the literature that were acquired with fiber-optic void fraction probes. Supported by the ONR under contract N00014-05-1-0121 and by the Spanish Government through grant: DPI2011-28356-C03-02.

3:07PM D28.00005 Laminar jet impingement and hydraulic jump behavior on a superhydrophobic surface with isotropic slip , JULIE VANDERHOFF, JOSEPH PRINCE, DANIEL MAYNES, Brigham Young University — We present an analytical model describing laminar jet impingement and the resulting hydraulic jump on a flat horizontal superhydrophobic surface with uniform surface slip in all directions. Due to the relatively thin film dynamics associated with the growth of the laminar jet after impingement, the influence of slip on the fluid physics is significant. An analysis based on momentum considerations is presented that allows prediction of the relevant thin film parameters as a function of radial position from the impingement point, jet Reynolds number, and constant relative slip length of the surface. The hydraulic jump can be located as a function of the laminar jet characteristics and imposed downstream liquid depth. The results reveal that at a given radial location, for increasing slip, the boundary layer growth and thin film thickness decrease while the surface velocity of the thin film increases. Increasing slip length also leads to the formation of a hydraulic jump at increasing radial location. A prediction equation is formulated to estimate the location of the hydraulic jump as a function of the magnitude of the slip and all other influencing variables.

3:20PM D28.00006 Computational Study of Air Entrainment by Plunging Jets – Influence of Jet Inclination , SURAJ DESHPANDE, MARIO TRUJILLO, University of Wisconsin - Madison — The process of air entrainment by a continuous liquid jet plunging into a quiescent liquid pool is studied computationally. Our earlier study [APS2011] focused on shallow impacts and the discernible periodicity of air cavity formation. Here, we consider the effect of jet angle. For steep impacts, we see a chaotic formation of small cavities, in agreement with the literature. To explain the difference, we track evolution of the flow from initial impact to quasi-stationary state, for different jet inclinations. The initial impact always yields a large air cavity, regardless of jet angle. Difference emerges in the quasi-stationary state where shallow jets demonstrate the periodicity but the steep jets do not. We show that this is a manifestation of the air entrainment being a function of flow disturbance. For shallow jets, the disturbance originates from strong wavelike motion of the cavity which results in a total disruption of the jet. Thus, the resulting cavities are large and occur periodically. For the steep jets, entrainment happens by collapse of a thin gas film uniformly enshrouding the submerged jet. Such a thin film is very sensitive to the local flow disturbances. Thus, its collapse occurs stochastically all around the jet causing chaotic entrainment of small air pocket.

3:33PM D28.00007 Laminar Jet Impingement and Hydraulic Jump Behavior on a Horizontal Surface with Anisotropic Slip , JOSEPH PRINCE, MICHAEL JOHNSON, JULIE VANDERHOFF, DANIEL MAYNES, Brigham Young University — We present an analytical model that describes the influence anisotropic slip exerts on the thin film and hydraulic jump dynamics of laminar jet impingement on horizontal surfaces. Superhydrophobic surfaces with alternating microscale ribs and cavities exhibit anisotropic slip in the azimuthal direction and thus are described by this model. The thin film dynamics are predicted by an integral momentum based analysis as a function of the jet Reynolds number and for a specified slip length that varies azimuthally. In the analysis the thickness of the thin film at a given radius is assumed to be independent of the azimuthal coordinate. The model shows that the boundary layer grows more slowly parallel to the ribs compared to other directions. A second momentum balance was performed that predicts the radial location of the hydraulic jump as a function of imposed downstream depth. Deviation from the classical no slip case and from the scenario of isotropic slip was determined over a range of possible slip lengths. The results show that the hydraulic jump radius in the direction parallel to the ribs is larger than in the transverse direction and the shape of the hydraulic jump is nearly elliptical. Comparisons between the model results and experimental measurements are also provided.

3:46PM D28.00008 Flow visualization of the water impact problem , HANS MAYER, ROUSLAN KRECHETNIKOV, University of California at Santa Barbara — When a flat plate impacts the surface of an incompressible viscous liquid, the liquid directly beneath the plate is set into motion and an ejecta – a high speed jet – forms at the plate edge giving rise to the familiar "splashing" behavior. We present the results of our experimental investigation of the water impact problem using a particle image velocimetry (PIV) system to quantify the flow field beneath the plate immediately after impact with the speeds of the order of $1\,\mathrm{m/s}$. The early-time formation of the ejecta for this flat plate geometry, including the influences of liquid viscosity ($1 < \mu < 10\,\mathrm{mPa \cdot s}$) and surface tension ($20 < \sigma < 70\,\mathrm{mN/m}$), are also studied with the PIV and high speed photography. Quantitative results for the flow field in the region beneath the plate and the growth of the ejecta are compared to existing and newly-developed theories.

3:59PM D28.00009 Origin of ejecta in the water impact problem , ROUSLAN KRECHETNIKOV, University of California at Santa Barbara — In this work we present a new analysis of the early time evolution of ejecta – jet forming during the impact of a flat plate on the surface of an incompressible viscous liquid. The key goals here are to clarify the effects of viscosity and surface tension. In the course of construction of the solution, first the standard assumptions behind the existence of the inviscid approximation are revisited. Second, scalings of the structure of the solution near the plate edge are determined, with which the viscous solution in the Stokes approximation near the edge is constructed analytically. Third, the structure of a uniformly valid solution matching the Stokes solution to the inviscid one is revealed here. Finally, the analysis of both viscous and inviscid solutions allows us to uncover the scalings for the early time-evolution of the ejecta.

4:12PM D28.00010 Three-dimensional oblique water-entry problems at small deadrise angles , MATTHEW MOORE, SAM HOWISON, JOHN OCKENDON, JAMES OLIVER, University of Oxford — We extend two-dimensional Wagner theory for the ideal, incompressible normal impact of rigid bodies that are nearly parallel to the surface of a liquid half-space. In particular, the impactors we consider are three-dimensional and have an oblique impact velocity that is an order of magnitude larger than the normal impact velocity of the body. A reformulation of the leading-order problem in terms of the displacement potential reveals the relationship between the oblique and corresponding normal impact solutions. We exploit this relationship to find new solutions to three-dimensional oblique impact problems. In the particular example of axisymmetric impactors, we consider several oblique-impact geometries in which singularities can develop on the boundary of the wetted region. We present the corresponding pressure profiles and models for the splash sheets and discuss the consequences of this breakdown of the theory.

Sunday, November 18, 2012 2:15PM - 4:25PM –

Session D29 Porous Media Flows II 32B - Chair: Dimitrios Papavassiliou, University of Oklahoma

2:15PM D29.00001 Distribution of flow-induced stresses in the pore space of random porous media[1]

, DIMITRIOS PAPAVASSILIOU, NGOC PHAM, University of Oklahoma — The distributions of stresses in the pore space of packed-sphere beds when a fluid flows through the porous medium under Darcy flow conditions are numerically computed with Lattice Boltzmann simulations. Three different ideally-packed (i.e., face-centered cubic, body-centered cubic and simple cubic packing) and one randomly-packed configuration of the packed bed are considered. It is found that the probability density function of the stresses, when the stresses are normalized with the mean value and the standard deviation of the distribution, behaves in different modes when the packing type changes. In the Darcy regime, the normalized stress distribution of a particular packing type is independent of the pressure difference and presents a unique pattern. The most important finding is that a log-normal distribution can successfully fit the stress distributions in the simulated randomly packed beds with high statistical accuracy. The applicability of the log-normal distribution is also explored for other types of porous media, and it is found that it is likely applicable for porous media with random pore space configurations.

[1]XSEDE under CTS090025

2:28PM D29.00002 The porous medium permeability and effective diffusion coefficient direct correlation

, BOJAN MARKICEVIC, Pall Corp, Cortland, NY — Dimensionless analysis of a momentum and mass transport in the homogeneous porous medium reveals that the permeability and effective to the molecular diffusion coefficient ratio can be expressed as a function of medium pore and throat sizes and two additional geometrical scales. These two scales, each one pertinent to the momentum and mass transport, respectively, are referred to as permeability and diffusivity characteristic scales. Based on these findings, it can be shown that the medium permeability and effective diffusivity can be correlated, and, at the same time, that one microscopic scale needs to be known in this correlation. The same is implied from the Katz-Thompson formula - which correlates the permeability, effective diffusivity, and breakthrough capillary pressure length scale. We recast the correlation developed into the Katz-Thompson formula form, showing how corresponding members are related. It turns out that the coefficient from the Katz-Thompson formula is equal to the ratio of the permeability to diffusivity characteristic length scales, and it is indeed constant for the homogeneous media. As porous media are heterogeneous materials, the analysis is extended onto such materials using heterogeneous capillary networks. The networks with the uniform, normal and *log*-normal pore size distribution functions are generated, where the networks are sufficiently large to obtain small variations in permeability and effective diffusivity for pore size distribution set. For such stochastically homogeneous media, the effective pore size averages are used in calculating the permeability and effective diffusivity showing the true nature of the coefficient in the Katz-Thompson formula.

2:41PM D29.00003 Capillary pinning of immiscible gravity currents in porous media

, BENZHONG ZHAO, Massachusetts Institute of Technology, CHRISTOPHER MACMINN, Yale University, MICHAEL SZULCZEWSKI, Massachusetts Institute of Technology, HERBERT HUPPERT, University of Cambridge, RUBEN JUANES, Massachusetts Institute of Technology — Gravity currents in porous media have attracted much interest recently in the context of geological carbon dioxide (CO2) storage, where supercritical CO2 is injected underground into deep saline aquifers. Capillary effects can be very important in the spreading and migration of the buoyant CO2 after injection because the typical pore size is very small ($\sim$10-100 μm), but the impact of capillarity on these flows is not well understood. Here, we study the impact of capillarity on a finite-release gravity current of a buoyant non-wetting fluid. Via simple, table-top experiments, we show that capillary pressure hysteresis causes pinning of a portion of the initial interface, which ultimately stops the spreading of the buoyant current at a finite distance. In addition, capillarity causes blunting at the leading edge of the draining buoyant current. We demonstrate through micromodel experiments that the height of the nose of the current is controlled by the pore geometry as well as the balance between capillarity and gravity. Our analysis suggests that capillary pinning and capillary blunting exert a fundamental control on the interface evolution of immiscible finite-release gravity currents in the context of CO2 sequestration in deep saline aquifers.

2:54PM D29.00004 Fluid Drainage from Porous Reservoirs[1]

, ZHONG ZHENG, Department of Mechanical & Aerospace Engineering, Princeton University, BEATRICE SOH, Department of Chemical & Biological Engineering, Princeton University, HERBERT HUPPERT, Institute of Theoretical Geophysics, Department of Applied Mathematics and Theoretical Physics, University of Cambridge, HOWARD STONE, Department of Mechanical & Aerospace Engineering, Princeton University, STONE GROUP TEAM — We report theoretical and experimental studies to describe buoyancy-driven fluid drainage from a porous medium. We first study homogeneous porous systems. To investigate the influence of heterogeneities, we consider the case where the permeability varies transverse to the flow direction, exemplified by a V-shaped Hele-Shaw cell. Finally, we analyze a model where both the permeability and the porosity vary transverse to the flow direction. In each case, a self-similar solution for the shape of the gravity current is found and a power-law behavior in time is derived for the mass remaining in the system. Laboratory experiments are conducted in homogeneous and V-shaped Hele-Shaw cells, and the measured profile shapes and the mass remaining in the cells agree well with our model predictions. Our study provides new insights into drainage processes such as may occur in a variety of natural and industrial activities including the geological storage of carbon dioxide.

[1]This work is supported by a grant from Carbon Mitigation Initiative at Princeton University

3:07PM D29.00005 A Solutal Fingering Instability during Capillary Imbibition in Porous Media

, W.D. RISTENPART, N.J. YOUNG, C.J. GUIDO, Dept. Chemical Engineering and Materials Science, Univ. 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 demonstrate that fingering instabilities occur with a wide variety of solutes and paper types, and we propose that the instability is driven by solute-induced changes in the air/liquid interfacial tension as liquid is absorbed into a humidity-dependent precursor film.

3:20PM D29.00006 ABSTRACT WITHDRAWN –

3:33PM D29.00007 ABSTRACT WITHDRAWN –

3:46PM D29.00008 Scaling laws for the imbibition of textured surfaces comprising short pillars with rounded edges

, KO OKUMURA, NORIKO OBARA, MINAKO HAMAMOTO-KUROSAKI, Ochanomizu University — Imbibition of porous media has been useful in many practical applications, e.g. to realize ultra-slippery surf surfaces. However, fundamental physical understandings are still limited. Recently, two scaling regimes are identified for the imbibition of textured surfaces comprising long pillars with sharp edges [1,2]. Here, we study textured surfaces comprising short pillars with rounded edges [3]. As a result, we find different scaling regimes for the dynamics. Surprisingly, this law is universal in the sense that it is independent of texture geometry, i.e., of pillar height, pillar distances, and pillar radius.

[1] Chieko Ishino, Mathilde Reyssat, Etienne Reyssat, Ko Okumura and David Quere, Europhys. Lett. 79 (2007) 56005.
[2] Minako HAMAMOTO-KUROSAKI and Ko OKUMURA, Eur. Phys. J. E 30, 283-290 (2009)
[3] Noriko OBARA and Ko OKUMURA, Phys. Rev. E Rapid Communication (in press 2012).

3:59PM D29.00009 Imbibition in porous media , BERTRAND LEVACHE, DENIS BARTOLO, Laboratoire PMMH ESPCI ParisTech, CNRS UMR 7636, MAURICE BOURREL, Total Petrochemical France, Pôle d'Etudes et de Recherche de Lacq — We study imbibition of wetting fluid into a uniform porous media made using microfluidic's techniques. We first revisit each phase of the diagram proposed by Lenormand. We then identify an invasion process that has not been seen until now where the invading fluid progress as thin films along the walls of the media. This modifies the problem by changing flows from two to three dimensions. Finally, we focus on this new phase, giving clues for the understanding of its dynamics and characterize it from a morphological point of view.

4:12PM D29.00010 Oil drainage in model porous media by viscoelastic fluids , JULIEN BEAUMONT, HUGUES BODIGUEL, ANNIE COLIN, Laboratoire du Futur (UMR 5258) — Crude oil recovery efficiency has been shown to depend directly on the capillary number (Ca). If the capillary phenomenon is well described for Newtonian fluids, the consequences of non linear rheology and viscoelasticity require more experimental work at the pore scale. In this work we take advantage of microfluidic to revisit this field. We carried out oil drainage experiments through a micromodel made up with photoresist resin. The wetting phase trapped is a model oil. The invading phases used are aqueous solutions of high molecular weight hydrolyzed polyacrylamide (HPAM) and surfactant. Qualitatively, we observed a transition between a capillary fingering at low flow rates and a stable front at high flow rates for the drainage experiments with HPAM and surfactant solutions as it happened for drainage with Newtonian fluids. From movies of the filling of the device, we determine the local velocity of all menisci in the porous media. Thus, we quantify the capillary fingering. Surprisingly, local velocities are not significantly different from those measured using water, whereas the HPAM solutions are much more viscous. With betaine solutions, we observed an emulsification of the oil clusters trapped during the invasion leading to a very high oil recovery after percolation.

Sunday, November 18, 2012 2:15PM - 4:25PM –

Session D30 Nanofluids: Computations I 33A - Chair: Nikolai Priezjev, Michigan State University

2:15PM D30.00001 Controlling Surface Roughness to Enhance Mass Flow Rates in Nanochannels[1] , MALGORZATA ZIMON, University of Strathclyde and STFC Daresbury Laboratory, DAVID EMERSON, STFC Daresbury Laboratory, JASON REESE, University of Strathclyde — A very active field of research in fluid mechanics and material science is predicting the behavior of Newtonian fluids flowing over porous media with different wettabilities. Opposite effects have been observed: some state that wall roughness always suppresses fluid-slip, whereas others show that for some cases roughness may reduce the surface friction. In this work, MD simulations were carried out to further investigate physical mechanisms for liquid slip, and factors affecting it. A rough wall was formed by either periodically spaced rectangular protrusions or was represented by a cosine wave. The MD simulations were conducted to study Poiseuille and Couette flow of liquid argon in a nanochannel with hydrophilic kryptonian walls. The effect of wall roughness and interface wettability on the streaming velocity, and the slip-length at the walls, is observed to be significant. Our results show a dependency of mass flow rate on the type of flow and topography of the channel walls. For a fixed magnitude of the driving force, an increase in the mass flow rate, compared to the smooth surface, was observed for the wavy roughness, whereas the opposite effect was observed for Couette flow where a higher slip was obtained for rectangular gaps.

[1]The study is funded in the UK by the Engineering and Physical Sciences Research Council.

2:28PM D30.00002 Properties of Water/Gold Nanofluids , GIANLUCA PULITI, SAMUEL PAOLUCCI, MIHIR SEN, University of Notre Dame — Nanofluids are believed to have enhanced thermophysical properties and heat transfer performance. The novelty of this work is in a comprehensive approach to understanding nanofluids through the use of molecular dynamics simulations with accurate potentials to model realistic materials. Specifically, this study treats the case of water confined between gold nanolayers to examine interfacial interactions and a water-based nanofluid with spherical gold nanoparticles. Properties are discussed for both systems. While the thermodynamic properties of the mixture are typically predicted using ideal mixture theory, such predictions are found to be generally poor for nanofluids. The anisotropy induced by the gold-water interface, and its effects, appear to be responsible for the disagreement. Transport properties, show a much more peculiar trend: the presence of nanoparticles has no effect on self-diffusion while viscosity increases drastically with respect to the prediction of classical theories. It is interesting to note that the thermal conductivity of nanofluids is enhanced for low particle concentrations, but it is below the prediction of classical theories for higher volume fractions. Interfacial effects appear, once again, to be responsible for such trend in transport properties.

2:41PM D30.00003 Nitrogen Flow in a Nanonozzle with Heat Addition , SERGEY AVERKIN, ZETIAN ZHANG, NIKOLAOS GATSONIS, Worcester Polytechnic Institute — The nitrogen flow in conical nanonozzles at atmospheric pressures are investigated using a three-dimensional unstructured direct simulation Monte Carlo (U3DSMC) method. The DSMC simulations are performed in computational domains that feature the plenum, the nanonozzle region and the external plume expansion region. The inlet and outlet boundaries are modeled by the Kinetic-Moment (KM) boundary conditions method. This methodology is based on the local one dimensional inviscid (LODI) formulation used in compressible (continuous) flow computations. The cross section for elastic collisions is based on the variable hard sphere (VHS) model. The Larsen–Borgnakke (L-B) model is used to simulate the exchange of the internal energy in the collision pair. Solid surfaces are modeled as being either diffuse or specularly reflecting. The effects of Knudsen number, aspect ratio, and nanonozzle scale on the heat transfer are investigating by ranging the throat diameters from 100-500 nm, exit diameter from 100-1000 nm, stagnation pressure from 1-10atm, and wall temperature from 300K-500K. Finite backpressure and vacuum conditions are considered. Macroscopic flow variables are obtained and compared with continuum predictions in order to elucidate the impacts of nanoscale.

2:54PM D30.00004 Molecular Dynamics Simulation of Gas Separation using Nanoporous Graphene[1] , HAROLD AU, MICHAEL BOUTILIER, Mechanical Engineering Dept., MIT, PIETRO POESIO, Dept. of Mechanical and Industrial Engineering, Brescia University, Italy, NICOLAS HADJICONSTANTINOU, ROHIT KARNIK, Mechanical Engineering Dept., MIT — We present molecular dynamics simulations of gas transport through nanoporous graphene to evaluate the latter's potential as a gas-separation membrane. Due to their very small thickness, such membranes are expected to exhibit high permeance. Provided precise tuning of the pore sizes is possible, graphene membranes have the potential to combine high permeance with high selectivity through molecular size exclusion. In the present study, we focus on separation of methane from hydrogen. Our results show that graphene with pores that are large compared to the kinetic diameters of both species are permeable to both gases. As pore size is reduced, we observe a greater decrease in the permeance of methane that results in a size exclusion effect for a range of pore sizes that are still permeable to hydrogen. Our results indicate that hydrogen permeance over this range of pores sizes is sufficiently large to offer considerable improvement compared to state of the art polymeric membranes.

[1]This work was supported, in part, by the MIT Energy Initiative Seed Fund Program

3:07PM D30.00005 Polarization as a field variable from molecular dynamics simulations , KRANTHI K. MANDADAPU, JEREMY TEMPLETON, JONATHAN LEE, Sandia National Laboratories — In this talk, we show that polarization density, an important quantity in electromagnetism, can be obtained from molecular dynamics simulations. We show that the Irving and Kirkwood procedure used for obtaining stresses and heat fluxes in terms of the microscopic quantities can be extended to the case of electrostatics where the macroscopic electrostatic equation can be derived starting with the microscopic electrostatic equation, microscopic density of charges and using a phase-space distribution function and a suitable localization function. As a result, we obtain an expression for polarization density as a field variable in terms of the microscopic dipole moments and quadrupole moments and higher order terms depending upon the degree of the polynomial used for the localization function. Finally, we apply this method to obtain the dielectric constant of bulk water and to study the polarization effects in electric double layer calculations. 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.

3:20PM D30.00006 Thermal Resistance and Temperature Jumps at Liquid/Solid Interfaces: Insights from Molecular Dynamics Simulations , M. PUCCI, S.M. TROIAN, California Institute of Technology, MC 128-95, Pasadena, CA 91125 — At macroscale dimensions, it is normally assumed that two distinct materials maintain equal temperature across the surface of contact. Even in the presence of a thermal flux across the interface, the contacting boundary is assumed to maintain thermal equilibrium so long as the interfacial resistance is negligible in comparison to that of the bulk. This has long been assumed an excellent approximation for liquid/solid interfaces, since liquids will conform in shape even to roughened surfaces. Recent molecular dynamics simulations of nanoscale films, however, have revealed the existence of intrinsic temperature jumps at liquid/solid interfaces. While previous studies have shown how stronger interaction energy between the liquid and solid will diminish temperature jumps, they cannot be altogether eliminated due to commensurability mismatch. Here we show a non-local effect in which the magnitude of the thermal jump is controlled by the thermal flux in the bulk. This finding suggests that temperature jumps across a liquid/solid interface are not simply a local effect due to the density mismatch across the interface. These jumps are also controlled by the rate of heat transfer, underscoring the importance of thermal resistance effects in nanoscale hydrodynamic systems.

3:33PM D30.00007 A multiscale method for modeling high-aspect-ratio micro/nano flows[1] , DUNCAN LOCKERBY, University of Warwick, MATTHEW BORG, JASON REESE, University of Strathclyde — In this paper we present a new multiscale scheme for simulating micro/nano flows of high aspect ratio in the flow direction, e.g. within long ducts, tubes, or channels, of varying section. The scheme consists of applying a simple hydrodynamic description over the entire domain, and allocating micro sub-domains in very small "slices" of the channel. Every micro element is a molecular dynamics simulation (or other appropriate model, e.g., a direct simulation Monte Carlo method for micro-channel gas flows) over the local height of the channel/tube. The number of micro elements as well as their streamwise position is chosen to resolve the geometrical features of the macro channel. While there is no direct communication between individual micro elements, coupling occurs via an iterative imposition of mass and momentum-flux conservation on the macro scale. The greater the streamwise scale of the geometry, the more significant is the computational speed-up when compared to a full MD simulation. We test our new multiscale method on the case of a converging/diverging nanochannel conveying a simple Lennard-Jones liquid. We validate the results from our simulations by comparing them to a full MD simulation of the same test case.

[1]Supported by EPSRC Programme Grant, EP/I011927/1.

3:46PM D30.00008 Influence of Slip Boundary Conditions and Confinement on Molecular Diffusion in Nanochannels: A Molecular Dynamics Simulation Study , ALI KHARAZMI, NIKOLAI PRIEZJEV, Michigan State University — We investigate the effects of confinement and slip boundary conditions on diffusion of solvent molecules in a nanochannel using the LAMMPS molecular dynamics program. In our simulations, the Lennard-Jones fluid is confined by crystalline substrates which allow fine adjustment of the slip length without changing the interfacial fluid structure. In the absence of flow, the molecular trajectories are used to compute the probability density of molecular displacements. The rate matrix that describes the time evolution of the probability density is then estimated by maximizing the likelihood function. Finally, the position-dependent diffusion coefficient is computed numerically from the Smoluchowski equation. We found that the local diffusion coefficient in the directions parallel and perpendicular to confining walls is a function of the distance from the confining walls and the degree of slip. These results are discussed in the context of nano-PIV (Particle Image Velocimetry) measurements of slip flows in nanochannels.

3:59PM D30.00009 Molecular dynamics simulations of oscillatory Couette flows with slip boundary conditions , NIKOLAI PRIEZJEV, Michigan State University — The effect of interfacial slip on steady-state and time-periodic flows of monatomic liquids is investigated using non-equilibrium molecular dynamics simulations. The simulations were performed in a wide range of oscillation frequencies; namely, when the Stokes boundary layer thickness is smaller than the channel width at the highest frequency, and, on the other hand, at lower frequencies that correspond to quasi-steady flows. It was found that the velocity profiles computed in MD simulations are well described by the continuum solution with the slip length as a fitting parameter that depends on the local shear rate. Interestingly, the shear rate dependence of the slip length obtained in steady-state shear flows is reproduced in oscillatory flows when the slip length is measured as a function of the absolute value of the local shear rate. Finally, for both types of flows, the friction coefficient at the liquid-solid interface correlates well with the structure factor and the contact density of the first fluid layer. Financial support from the National Science Foundation (CBET-1033662) is gratefully acknowledged.

4:12PM D30.00010 Effective Translational Diffusion of Nanorotors as Rotary Powered Random Walkers , AMIR NOURHANI, PAUL LAMMERT, Phys. Dpt., Penn State Univ., ALI BORHAN, Chem. Eng. Dpt., Penn State Univ, VINCENT CRESPI, Phys. Dpt., Penn State Univ. — The coupling of the orientational stochastic dynamics and rotary powered dynamics at different dimensions leads to an effective translational diffusion of a rotary powered random walker. In a conventional nanorotor system, moving in two-dimension close to a substrate, the one-dimensional orientational stochastic dynamics couples to the rotary deterministic motion and leads to an effective two-dimensional translational diffusion, which is chiral in short to medium time scales. If a nanorotor can have three-dimensional dynamics, an emergent three-dimensional effective diffusion would be the outcome of the coupling between three one-dimensional orientational stochastic processes and a two-dimensional deterministic rotation in the plane of motion. Such effective diffusion processes are a property of nanoscale where the deterministic and stochastic dynamics are both significant.

Sunday, November 18, 2012 2:15PM - 4:25PM –

Session D32 Focus Session: Vortex Dynamics in Fluid-Structure Interactions II 33C - Chair: Darren Crowdy, Imperial College London

2:15PM D32.00001 Analytical solutions for hollow vortex pairs in a channel , CHRISTOPHER GREEN, DARREN CROWDY, Imperial College London — Motivated by the problem of vortex rings travelling along tubes, we study the two-dimensional analogue comprising a counter-rotating vortex pair travelling along a straight-walled channel. By modelling the vortices as a pair of so-called hollow vortices we are able to find a class of solution in closed mathematical form. Solutions for both a single vortex pair and a periodic array of vortex pairs will be presented. Connections with classical solutions due to both Michel (1890) and Pocklington (1895) will be made.

2:28PM D32.00002 Hollow Vortices in Flow Past a Flat Plate , ALAN ELCRAT, Wichita State University, LUCA ZANNETTI, Politecnio di Torino — Closed and open hollow wakes are considered as analytic models for the 2D inviscid steady flow past a plate normal to the stream. It is shown that only open configurations which satisfy the Kutta condition exist. The main argument is based on considering a plate located on the edge of a step with varying height. It is shown that solutions for open wakes exist for backward, null and forward-facing steps, while closed wakes only exist for backward-facing steps. The occurrence of secondary separation has been modeled by adding a hollow region attached to the downstream corner. Peculiar accuracy issues of the problem are pointed out which may explain other contradictory results from the literature. It is shown how the Kirchhoff wake is a limiting solution for certain values of the governing parameters.

2:41PM D32.00003 Vortex Dynamics at Early Time Stages of Viscous Flow past a Finite Plate[1] , LING XU, MONIKA NITSCHE, University of New Mexico — We use numerical simulations to revisit a fundamental problem of viscous flow past a finite flat plate. We resolved the boundary layer separation and roll-up from very early time to relatively large times. Details of vorticity structure in the boundary layer at early times are shown, these features have not been studied before. In particular, we will present details of the negative vorticity region, the entrainment between the positive and negative vorticity and the scales of circulation shedding rate from the plate tip, maximum velocity and core vorticity.

[1]Xsede

2:54PM D32.00004 Frequency spectrum and scale dependence of a propulsive self-excited vortex generator[1] , ROBERT WHITTLESEY, JOHN DABIRI, California Institute of Technology — We describe the development and characterization of a passive device that creates a train of vortex rings from a steady incoming flow. The device consists of a collapsible tube enclosed in an air-tight chamber which undergoes self-excited oscillations under specific conditions of flow rate and transmural pressure. An experimental parameter study was conducted in order to determine the oscillation frequency spectrum of the device, and its dependence on the nozzle diameter. For certain combinations of flow rate and transmural pressure, the frequency of self-excited oscillations, and hence vortex formation, is independent of device size over an order of magnitude range of device volume. These results have a particular interest for the development of vehicles utilizing vortex-enhanced propulsion (Ruiz et al, JFM, 2011). Continued work in this area has focused on the implementation of this device to a self-propelled submarine.

[1]Support is greatly appreciated from ONR Awards N000140810918 and N000141010137.

3:07PM D32.00005 Three-Dimensional Vortex Design by Hydrofoil Acceleration , MARTIN SCHEELER, DUSTIN KLECKNER, WILLIAM IRVINE, University of Chicago — We demonstrate the use of accelerated hydrofoils (airfoils in water) for the generation of vortex loops of arbitrary shape in three dimensions. The technique allows not only the patterning of shape, but non-trivial topology. We study both the process of vortex production and the subsequent vortex evolution using ultra-fast 3D laser scanning tomography.

3:20PM D32.00006 An Experimental Investigation on the Interference of the Multiple Wind Turbines with Different Layout Patterns in Atmospheric Boundary Layer Winds[1] , HUI HU, WEI TIAN, AHMET OZBAY, Iowa State University — We report an experimental study to investigate the wake interferences of multiple wind turbines in atmospheric boundary layer (ABL) winds. The experimental study is conducted by taking advantages of the large-scale Aerodynamic/Atmospheric Boundary Layer (AABL) Wind Tunnel available at Iowa State University to quantify the performances of an array of wind turbine models with aligned and staggered arrangement patterns. In addition to measuring dynamic wind loads (both forces and moments) and the power outputs of the wind turbine models, advanced flow diagnostic techniques such as digital Particle Image Velocimetry (PIV) is used to conduct detailed flow field measurements to quantify the flow characteristics of the surface winds and wake interferences among the multiple wind turbines with different layout patterns. The detailed flow field measurements are correlated with the dynamic wind loads and power output measurements to elucidate underlying physics for the optimal design of the wind turbine array layout with the ultimate goal of higher total power yield and better durability of the wind turbines operating in more realistic environments.

[1]The reserach work is funded by IAWIND and NSF with Award # CBET-1133751.

3:33PM D32.00007 Von Kármán vortex streets: analytical solutions with distributed vorticity , DARREN CROWDY, CHRISTOPHER GREEN, Imperial College London — We present analytical solutions for both staggered and unstaggered von Kármán vortex streets. Instead of point vortices, we employ a distributed vorticity model consisting of arrays of hollow vortices whose shapes are determined as part of the solution scheme (it is a free boundary problem). The new solutions are compared with purely numerical solutions based on the vortex patch model computed by Saffman & Schatzman in 1981.

3:46PM D32.00008 Structure and stability of the finite-area von Karman street , P. LUZZATTO-FEGIZ, University of Cambridge, C.H.K. WILLIAMSON, Cornell University — By using a recently developed numerical method, we explore in detail the equilibria for a Karman street of uniform, large-area vortices. We construct a reliable implementation of an energy argument to find superharmonic instabilities. This leads us to organize flows into families with fixed impulse I, and to construct diagrams of the flow energy E and horizontal spacing L. Families of large-I streets exhibit a turning point in L, and terminate with "cat's eyes" vortices (as also suggested by previous investigators). However, for low-I streets, the solution families display a multitude of turning points (leading to multiple possible streets, for given L), and terminate with teardrop-shaped vortices. This is radically different from previous suggestions in the literature. These two qualitatively different limiting states are connected by a special street, whereby vortices from opposite rows touch, such that each vortex exhibits three corners. Furthermore, by following the family of $I = 0$ streets to small L, we access a large, hitherto unexplored regime, involving streets with L much smaller than previously believed possible. For each solution family, our stability approach also reveals a single superharmonic bifurcation, leading to new vortex streets, which exhibit lower symmetry.

3:59PM D32.00009 Nonholonomic Mechanics and Fluid-Body Interactions , SCOTT KELLY, PHANINDRA TALLAPRAGADA, University of North Carolina at Charlotte — Certain velocity constraints arising in idealized models for fluid-body interactions, including the Kutta condition classically applied at the trailing cusp of a Joukowski hydrofoil experiencing lift, are formally equivalent to nonintegrable constraints arising in the mechanics of finite-dimensional mechanical systems. This equivalence allows hydrodynamic problems involving vortex shedding and related phenomena to be framed in the context of geometric mechanics on manifolds, and for essential mechanisms of biomorphic aquatic propulsion to be interpreted in terms of symmetry-breaking and generalized momentum equations. We illustrate this perspective using simple examples that highlight parallels between the finite- and infinite-dimensional cases.

4:12PM D32.00010 Fluid structure interaction with low and high order flexibility using volume penalization , JIANXIONG SHENG, Institute for Atmospheric and Climate Science, ETH, Zurich, Switzerland, THOMAS ENGELS, M2P2-CNRS, Aix-Marseille University, Marseille, France & Institut fuer Stroemungmechanik und Technische Akustik (ISTA), TU Berlin, Germany, DMITRY KOLOMENSKIY, Centre Europeen de Recherche et de Formation Avancee en Calcul Scientifique (CERFACS), Toulouse, France, KAI SCHNEIDER, M2P2-CNRS & CMI Aix-Marseille University, Marseille, France — We present a new numerical scheme for the simulation of deformable objects immersed in a viscous incompressible fluid. The two-dimensional Navier–Stokes equations are discretized with an efficient 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 solid object. The code is validated using classical fluid-structure interaction benchmarks, a channel flow with an immersed cylinder and attached flexible foil. We make a comparison between numerical simulations of deformable foils of two different types. The first consists of two rigid plates linked with a torsion spring (low-order flexibility). The second is a flexible plate modeled using a non-linear beam equation (high-order flexibility). We also compare these results with numerical simulations and experiments carried out by Toomey and Eldredge (Phys. Fluids 20, 073603, 2008).

Sunday, November 18, 2012 4:45PM - 5:50PM –

Session E1 Geophysical: Ocean I 22 - Chair: S. Balachandar, University of Florida

4:45PM E1.00001 Quantifying Suspended Sediment Diffusion Through Direct *in situ* Measurements of Turbulent Schmidt Number , IAN TSE, EVAN VARIANO, University of California, Berkeley — In this study we investigate how the diffusion of suspended sediment differs from the diffusion of fluid momentum using both laboratory and *in situ* field measurements. The most common model for turbulent diffusion considers eddy diffusivity (D_T) to be proportional to the eddy viscosity (ν_T) scaled by the turbulent Schmidt number (σ_T). But accurate selection of σ_T values is challenging because sediment, by virtue of its inertia, is transported differently than either momentum or passive scalars. We directly measure σ_T over a variety of flow cases using a novel Volumetric Particle Imager (VoPI) developed for this purpose. VoPI is a field-deployable quantitative imaging device that can obtain three-component particle velocity records in a volume. By computing velocity variances and integral timescales from measured Lagrangian velocity records, we compute D_T (for particles) and ν_T (for tracers) directly using Taylor's (1921) formulation. We present the construction and calibration of the device as well as validation of its measurements. We also report the connections between the measured σ_T values and the flow conditions in which they occur and suggest predictive methods for when direct measurements are unavailable.

4:58PM E1.00002 High-Schmidt-number mass transport mechanisms from a turbulent flow to absorbing sediments , CARLO SCALO[1], UGO PIOMELLI, LEON BOEGMAN, Queens University — We have investigated the mechanisms involved in dissolved oxygen (DO) transfer from a turbulent flow to an underlying organic sediment bed, populated with DO-absorbing bacteria, relying on the coupling between the bio-geochemistry of the sediment layer and large-eddy simulation for the transport on the water side [Scalo et al., *J. Geophys. Res.*, *117*(C6), 2012]. Time correlations at the sediment-water interface (SWI) show that the diffusive sublayer acts as a de-noising filter with respect to the overlying turbulence; the mass flux is not affected by low-amplitude background fluctuations in the wall-shear stress but, rather, by energetic and coherent near-wall transport events, in agreement with the surface renewal theory. The spatial and temporal distribution of the mass flux is therefore modulated by rapidly evolving near-wall high-speed streaks (associated with intermittent peaks in the wall-shear stress) transporting patches of (rich-in-oxygen) fluid to the edge of the diffusive sublayer, leaving slowly-regenerating elongated patches of positive DO concentration fluctuation and mass flux at the SWI. The sediment surface retains the signature of the overlying turbulent transport over long time scales, allowed by the slow bacterial absorption.

[1]Currently postdoctoral fellow at Center for Turbulence Research (scalo@stanford.edu)

5:11PM E1.00003 Sedimentation of porous and solid particles in stratified fluids[1] , SHILPA KHATRI, CAROL ARNOSTI, ROBERTO CAMASSA, CLAUDIA FALCON, XIE HE, RICHARD MCLAUGHLIN, JENNIFER PRAIRIE, BRIAN WHITE, SUNGDUK YU, KAI ZIERVOGEL, University of North Carolina at Chapel Hill, Mathematics and Marine Sciences — Marine aggregates, particles composed of organic and inorganic material in the ocean, are fundamental to marine carbon cycling both in their importance to bacterial remineralization and carbon flux from the surface ocean. Understanding the function of marine aggregates in carbon biogeochemistry requires knowledge of their small scale settling dynamics in different physical environments. We have conducted experiments to study the settling behavior of single solid and porous spheres and natural marine aggregates through sharp vertical density stratification in ambient fluids. Additionally, we have investigated the behavior of particle clouds. In all of these situations, particles demonstrate decreased settling velocity at the density transition which could be brought about by entrainment of less dense fluid from above and/or diffusion-limited retention. By comparing experimental results to models including entrainment and diffusion, we have identified the mechanisms underlying this delayed settling phenomenon. Discussion of the models will be presented.

[1]This research is supported by NSF RTG DMS-0502266, NSF RTG DMS-0943851, NSF RAPID CBET-1045653, NSF CMG ARC-1025523, NSF DMS-1009750, and ONR DURIP N00014-09-1-0840.

5:24PM E1.00004 Numerical simulation on fine sediment transport in steady and oscillatory boundary layer – The role of rheology , XIAO YU, University of Delaware, EMRE OZDEMIR, WHOI, TIANJIAN HSU, University of Delaware, SIVARAMAKRISHNAN BALACHANDAR, University of Florida — Turbulence-resolving 3D numerical simulations of fine sediment transport in both steady and oscillatory boundary layers are carried out to study the interplay between turbulence modulation and rheological stress. A high-accurate scheme is developed to resolve all the scales of carrier flow turbulence. Fourier expansions are adopted in both streamwise and spanwise directions. To incorporate both the hindered settling effect and rheology models, a sixth-order compact finite difference scheme is implemented in vertical direction to keep the spectral-like accuracy. A recent numerical study (Ozdemir et al. 2010, J. Fluid Mech.) on fine sediment transport in the wave boundary layer reveals the evolution of transport regimes from a well-mixed sediment distribution, to the formation of lutocline and a complete laminarization of wave boundary layer due to increasing sediment availability and settling velocity. We are motivated to further study the effect of rheology in determining the transition of flow regimes and hydrodynamic dissipation. By including the rheology, simulation results shows that the increased effective viscosity tends to increase the thickness of viscous sub-layer and further reduce the thickness of turbulent regime, which is limited by the lutocline.

5:37PM E1.00005 Mechanisms of complete turbulence suppression in turbidity currents[1] , MRUGESH SHRINGARPURE, Department of Mechanical and Aerospace Engineering, University of Florida, Gainesville, Florida, USA, MARIANO CANTERO, Institute Balseiro, (CNEA-UNCu), Bariloche Atomic Center, San Carlos de Bariloche, Rio Negro, Argentina, S. BALACHANDAR, Department of Mechanical and Aerospace Engineering, University of Florida, Gainesville, Florida, USA — The sustained propagation of turbidity current depends on a tight interplay between suspended sediments and turbulence. This work explores the phenomenon of complete turbulence suppression in a dilute turbidity current due to stratification of suspended sediments. Direct numerical simulations of turbidity currents are carried out to understand the dynamics of complete turbulence suppression. We observe that stratification of sediments leads to damping and spatial redistribution of hairpin and quasi-streamwise turbulent structures in the flow. These turbulent structures are known to be responsible for sustaining turbulence in the flow. We propose that beyond a critical stratification limit, the existing vortical structures in the flow are damped to an extent where they loose their ability to auto-generate subsequent turbulent structures, which ultimately leads to complete loss of turbulence. We also identify three parameters: Reynolds number (Re_τ), Richardson number (Ri_τ) and sediment settling velocity (V_z) to control the flow dynamics. Therefore a criteria for complete turbulence suppression can be defined as a critical value for $Ri_\tau V_z$. Based on simulations, experiments and field data, the critical value appears to have logarithmic dependence on Re_τ.

[1]Authors thank the support of NSF through grant OCE1131016.

Sunday, November 18, 2012 4:45PM - 5:50PM –

Session E2 Convection and Buoyancy-Driven Flows III: Rayleigh-Bénard Convection 23A - Chair: Yuri Feldman, California Institute of Technology

4:45PM E2.00001 Turbulent thermal convection with polymer additives and with smooth and rough top and bottom plates[1] , KE-QING XIA, PING WEI, RUI NI, XIAO-MING LI, Department of Physics, The Chinese University of Hong Kong — We present an experimental study of heat transport in turbulent Rayleigh-Bénard (RB) convection with polymer additives made in two convection cells, one with a smooth top and bottom plates and the other with a rough top and bottom plates. For the cell with smooth plates, a reduction of the measured Nusselt number (Nu) was observed. For the cell with rough plates, however, an enhancement ($\sim 4\%$) of Nu of was observed when the polymer concentration is greater than 120 ppm. This increase in Nu is corroborated by an increased large-scale circulation (LSC) velocity in the same cell when polymers are added. In contrast, the LSC velocity in the smooth cell is found to be essentially the same with and without polymers. Results from local velocity field and energy dissipation rate measurements will also be presented.

[1]This work was supported by RGC of Hong Kong SAR (Nos. CUHK404409 and CUHK403811).

4:58PM E2.00002 On Laminar and Turbulent Free Convection in Thin Spherical Shells[1] , YURI FELDMAN, TIM COLONIUS, California Institute of Technology, Pasadena, CA, USA, 91125 — Laminar and turbulent free convection flow inside thin spherical shells with isothermal cold and hot boundaries and internal/external radius ratios in the range of $0.85 \leq r_i/r_o \leq 0.95$ is numerically investigated. The accuracy of the results has been verified by grid independence analysis and DNS-LES comparisons of the flow characteristics for the typical cases. The functional *Nu-Ra* dependency is extensively investigated for the range of $10^3 \leq Ra \leq 10^{10}$ including laminar, transitional and fully turbulent flow regimes. For thin shells, we observe considerable deviations from the existing engineering correlations. The deviations tend to increase for transitional and fully turbulent flows. A new correlation for *Nu-Ra* dependency is proposed and favorably verified by independently obtained experimental end numerical results. The influence of non-uniform temperature distribution along the shell boundaries on the overall heat flux rate is also discussed.

[1]Part of this research was carried out at the Jet Propulsion Laboratory, California Institute of Technology, under a contract with the National Aeronautics and Space Administration.

5:11PM E2.00003 Natural convection in a partially heated cylindrical container , JOSE NUÑEZ, Universidad de la Cienega del Estado de Michoacan de Ocampo, EDUARDO RAMOS, Universidad Nacional Autonoma de Mexico — In this contribution, we study the stability of the natural convection flow in a vertical cylinder heated from below and cooled from above and with partial heating on the lateral wall with a numerical model. A mixed Fourier Galerkin- Finite volume method was used for the numerical integration. The results indicate that for small Rayleigh numbers the flow is axisymmetric with the motion confined to the upper part where the temperature gradients are concentrated. At Ra=1.5e+4 the axisymmetric flow loses stability to a non-axisymmetric motion with an azimuthal mode k=1, through a pitchfork bifurcation. Further increase in the Rayleigh number results in a time dependent flow with alternate flows with modes k=2, 3.

5:24PM E2.00004 Constant- flux discrete heating in a unit aspect-ratio annulus , JUAN M. LOPEZ, Arizona State University, SANKAR MANI, East Point College of Engineering and Technology, YOUNGHAE DO, Kyungpook National University, FRANCISCO MARQUES, Univ. Politècnica de Catalunya — Natural convection in an annulus with a discrete heat source on the inner cylinder is studied numerically. The outer cylinder is isothermally cooled at a fixed low temperature, and the top wall, the bottom wall and unheated portions of the inner cylinder are thermally insulated. For low applied heat flux through the heater, as measured non-dimensionally by a Grashof number, Gr, the flow in the annular gap consists of a weak single-cell overturning meridional flow and heat is transported primarily via conduction. As the nonlinear convection terms become more important, the meridional circulation sweeps the isotherms from being almost vertical near the outer cylinder to almost horizontal near the bottom wall. By the end of the transition from the conductiondominated regime ($Gr < 10^4$) to the convection-dominated regime ($Gr \sim 10^6$), the flow becomes segregated into three distinct regions and there is a strong wall plume originating at the heater that reaches the top and forms a large scale wavy structure along the top. For $Gr \sim 10^{10}$, this wavy structure becomes unstable to 3D instability with high azimuthal wavenumbers $m \sim 30$.

5:37PM E2.00005 Heat transfer through suspensions of particles in turbulent convection , ANDREA SCAGLIARINI, ARMANN GYLFASON, University of Reykjavik, FEDERICO TOSCHI, Eindhoven University of Technology — We study, by means of numerical simulations the turbulent dynamics of a layer of liquid confined between two plates, heated form below and cooled from above (as in the standard Rayleigh-Bénard setup), with suspended solid particles. We consider both the cases where particles exert or not a back-reaction on the fluid flow. In particular we will address the dependence of the heat transfer (quantified by the Nusselt number) on particles volume fraction, heat diffusivity and of particle properties.

Sunday, November 18, 2012 4:45PM - 5:24PM –

Session E3 Multiphase Flows: Boiling 23B - Chair: Peter Taborek, University of California Irvine

4:45PM E3.00001 Lattice Boltzmann Simulation of Thermal Multiphase Flows with Dynamic Wall Interactions , MICHAEL IKEDA, LAURA SCHAEFER, University of Pittsburgh — As energy densities in electronic devices rapidly increase, improved two-phase microchannel heat exchanger designs are of great interest. However, a better understanding of flow boiling in these regimes is required. The lattice Boltzmann method (LBM) has shown great promise in the simulation of multiphase flows due to its ability to easily capture interfacial dynamics. Although there have been many recent developments to the standard thermal, multiphase LBM, wall interactions are typically oversimplified. These simplifications lead to interactions which are only appropriate for isothermal, static simulations. In this work, we extend wall interactions based on the pseudopotential multiphase approach to include the variable wetting behavior that occurs with changing temperatures. This will enable the future modeling of the flow boiling process with temperature-dependent wetting characteristics.

4:58PM E3.00002 Computations of Nucleate Boiling[1] , GRETAR TRYGGVASON, University of Notre Dame, JIACAI LU, Worcester Polytechnic Institute — Simulations of boiling flows have progressed significantly in the last decade and it is now possible to accurately compute the film boiling of fluids under a wide range of conditions, for example. Although some progress has been made in simulating nucleate boiling, considerable challenges remain, particularly for water under atmospheric pressure. The challenges include the resolution of thin film between a growing vapor bubble and the hot wall, steep thermal gradient at the phase boundary, and the determination of the distribution and activation of nucleation sites. We report on recent progress using a front tracking method to follow the phase boundary, coupled with a multiscale strategy to capture the microlayer and resolve steep thermal gradients. The results including comparison with experimental results and simulations of bubbles released from multiple nucleation sites for both pool and flow boiling. Preliminary results indicate that for moderate nucleation site density the bubbles are formed relatively independently of each other.

[1]Research supported by DOE (CASL).

5:11PM E3.00003 Transient two-phase flow in microfluidics and nanofluidics , ANGEL VELASCO, ANDREW SONG, SERAH FRIEDMAN, MATTHEW PEVARNIK, ZUZANNA SIWY, PETER TABOREK, University of California, Irvine — We have studied the flow of a high pressure liquid (nitrogen and water) into vacuum through large aspect ratio pipes with diameters ranging from 25 microns to 50 nanometers. The decreasing pressure in the pipe induces boiling when the saturated vapor pressure is reached, creating a two-phase liquid/vapor flow. A novel method of measuring extremely small flow rates based on mass spectrometry will be presented. The validity of the method was verified using measurements of the flow of helium and argon through standard micron scale capillary tubes; subsequent measurements used single ion track pores which were 12 microns long with diameters in the range of 800-50 nm. A systematic study with nitrogen at 77 K was done with inlet pressures above and below the saturated vapor pressure. When the applied pressure is below the saturated vapor pressure the single phase flow was observed to obey the compressible Navier-Stokes equation. At pressures greater than the saturated vapor pressure, a stable flow was observed in pipes with diameters greater than 5 microns. For diameters below 2 microns significant fluctuations in the flow rate are observed at applied pressures up to 35 Atm, suggesting the onset of two-phase flow.

Sunday, November 18, 2012 4:45PM - 5:50PM –

Session E4 Drops III 23C - Chair: Julie Vanderhoff, Brigham Young University

4:45PM E4.00001 Influence of Flow on Longevity of Superhydrophobic Coatings , MOHAMED A. SAMAHA, HOOMAN VAHEDI TAFRESHI, MOHAMED GAD-EL-HAK, Department of Mechanical & Nuclear Engineering, Virginia Commonwealth University, Richmond, VA 23284 — Previous studies have demonstrated the capability of superhydrophobic surfaces to produce slip flow and drag reduction, which properties hold considerable promise for a broad range of applications. However, in order to implement such surfaces for practical utilizations, environmental factors such as water movement over the surface must be observed and understood. In this work, experiments were carried out to present a proof-of-concept study on the impact of flow on longevity of polystyrene fibrous coatings. The time-dependent hydrophobicity of a submerged coating in a pressure vessel was determined while exposing the coating to a rudimentary wall-jet flow. Rheological studies were also performed to determine the effect of the flow on drag reduction. The results show that the longevity of the surface deteriorates by increasing the flowrate. The flow appears to enhance the dissolution of air into water, which leads to a loss of drag-reduction.

4:58PM E4.00002 Use of the string method to find minimal energy paths of droplets on superhydrophobic surfaces , KELLEN PETERSEN, Courant Institute of Mathematical Science, New York University — Interest in superhydrophobic surfaces has increased due to interesting advances in science and engineering. Here we use a diffuse interface model for droplets on topographically and chemically patterned surfaces. We then apply the constrained string method to examine the transition of droplets between different metastable/stable states. The string method finds the minimal energy paths (MEPs) which correspond to the most probable transition pathways between the metastable/stable states in the configuration space. In the case of a hydrophobic surface we determine the MEP corresponding to the transition between the Cassie-Baxter and Wenzel states. Additionally, we realize critical droplet morphologies along the MEP associated with saddle points of the free-energy potential and the energy barrier of the free energy. We analyze and compare the MEPs and free-energy barriers for a variety of surface geometries, droplets sizes, and static contact angles ranging. We demonstrate the string method as a useful tool in the study of droplets on superhydrophobic surfaces by presenting a numerical study that finds MEPs in configuration space, critical droplet morphologies and free-energy barriers which in turn give us a greater understanding of the free-energy landscape.

5:11PM E4.00003 Characterization of heat transfer from superhydrophobic substrates to water droplets , ROBB HAYS, JULIE VANDERHOFF, DANIEL MAYNES, Brigham Young University — We report on measurements of thermal transport to solitary sessile water droplets placed on heated superhydrophobic substrates at constant temperature. Data was obtained by heating the substrates to specified constant temperatures and gently placing a single water droplet on the surface. The droplet was allowed to evaporate completely while two video cameras and one infrared camera captured images of the droplet. The images were post-processed to yield transient geometric and thermal information, including droplet volume, droplet-substrate contact area, and droplet temperature. The total evaporation time and transient and average convective heat transfer coefficients were determined from the measurements as a function of the substrate surface temperature and the superhydrophobic topography. Four different superhydrophobic surfaces were investigated: rib-patterned surfaces of 50%, 80%, and 93% cavity fraction and a post-patterned surface of 97% cavity fraction. Ribs and posts ranged in width from 3 to 30 μm and in height from 15 to 20 μm. Cavities between ribs or posts ranged in width from 30 to 37 μm. Results were also obtained for hydrophilic, smooth hydrophobic, and superhydrophilic substrates for comparison purposes. In general, the evaporation time and difference between the droplet and substrate temperatures are both much greater for the superhydrophobic surfaces compared to smooth surfaces.

5:24PM E4.00004 Two-Pronged Jet Formation Caused by Droplet Impact on Anisotropic Superhydrophobic Surfaces , JOHN PEARSON, DANIEL MAYNES, DAVID BILODEAU, BRENT WEBB, Brigham Young University — When a liquid droplet impacts a horizontal superhydrophobic surface with anisotropic surface patterning in the form of alternating ribs and cavities, the rebounding droplet can exhibit a unique two-pronged jet emission under certain conditions. This behavior occurs as a result of the unequal shear stresses and dynamic contact angles that exist along the two major axes: parallel and perpendicular to the ribs/cavities. Due to these unequal conditions in the two major directions, the droplet spread and collapse occurs more rapidly in the direction parallel to the ribs/cavities than the transverse direction. Droplet impact experiments with eleven different fluids of viscosity that varied by more than three orders of magnitude were conducted, and the ranges of Capillary number, *Ca*, and Ohnesorge number, *Oh*, over which the two-pronged phenomenon occurs have been quantified. Further, the probability of the two-pronged jet emission has been quantified as a function of *Oh* and Weber number, *We*. For $Oh > 0.0154$, the behavior was never observed, while at lower values of *Oh*, the behavior occurs for an intermediate range of *Ca* that depends on *Oh*.

5:37PM E4.00005 Spreading of liquid drop on superhydrophilic micropillar array , SEONG JIN KIM, MYOUNG-WOON MOON, KWANG-RYEOL LEE, Korea Institute of Science and Technology, HO-YOUNG KIM, Seoul National University — When a drop is deposited on a superhydrophilic micropillar array, the upper part of the drop (referred to as the bulk) collapses while the bottom part penetrates into the gaps of the array, forming a fringe film. Here we quantify the dynamics of this process using a combination of experiment and theory. In the early stages when the fringe extension is negligible compared to the bulk radius, both the spreading of the bulk and the entire drop footprint follow the same power law ($t^{1/4}$), t being time. The bulk shrinks toward the end of the spreading process due to the drainage of liquid into the fringe film. The film spreads like $t^{1/2}$ until the end of the process. A remarkable finding is that the entire footprint grows like $t^{1/4}$ despite the diffusive growth of the fringe film, implying that the shrinkage of the bulk compensates for the outward spreading of the film. We rationalize some of these results with scaling analyses based on the balance of capillary forces that drive the flow and viscous shear forces.

Sunday, November 18, 2012 4:45PM - 5:50PM –

Session E5 Computational Fluid Dynamics III 24A - Chair: Carlos Pantano, University of Illinois at Urbana-Champaign

4:45PM E5.00001 Simulation of Turbulence using Quasi Equilibrium Lattice Boltzmann Method , CHAKRADHAR THANTANAPALLY, DHIRAJ V. PATIL, Jawaharlal Nehru Centre for Adv Sci Research, India, SAURO SUCCI, Istituto Applicazioni Calcolo "Mauro Picone," C.N.R, Italy, SANTOSH ANSUMALI, Jawaharlal Nehru Centre for Adv Sci Research, India — Development of accurate and efficient methods for DNS of turbulence, where degrees of freedom associated with the flow scales with Reynolds number as $Re^{9/4}$, is one of the important goals of computational fluid dynamics. In this regard, Lattice Boltzmann method (LBM) is an attractive option due to high parallel scalability and its ease of application to complex geometries. Recently, it was shown that energy conserving LBM is superior over their athermal counterpart due to improved stability and increase in accuracy for high resolution simultations. However, in subgrid domain the behavior is found to be opposite. In this work, we show that via multi-relaxation time (MRT) model, it is possible to preserve the accuracy of the energy conserving LBM for both subgrid as well as high resolution simulation models. We show that introducing Prandtl number, as a means to subgrid viscosity, allows to do under resolved simulations quite efficiently and motivate this behavior via sound relaxation mechanism. The model showed to perform well over the regular thermal and athermal LBM at lower resolutions. The stability and accuracy of the model is validated using two-dimensional Taylor-Green and double periodic shear layer, and three-dimensional Kida-Pelz and Taylor-Green initial conditions.

4:58PM E5.00002 Hybrid Lattice-Boltzmann model for Thermally Coupled Fluid-Solid Problem , LEITAO CHEN, LAURA SCHAEFER, University of Pittsburgh — The most commonly used thermal boundary condition on solid wall in fluid problem is either type of Neumann or Dirichlet. However, the thermal boundary condition on solid wall in many practical problems is much more complicated and impossible to predict especially when the flow is unsteady or involves complex geometry such as porous media. So the best cure is to simulate fluid and solid together. Lattice-Boltzmann Method is becoming a promising alternative scheme for simulating thermal fluid flows while in the same time solving the conventional energy equation with Finite Volume method is still superior to other methods in modeling pure heat conduction in solid. In this work a 2D hybrid model is built, in which the traditional Lattice-Boltzmann BGK model on D2Q9 lattice for fluid part is coupled with the Finite Volume model on unstructured mesh for solid part. In addition, the numerical schemes on thermal fluid-solid interface for both straight and curved wall are developed. The Hybrid model is proved to be able to solve thermally coupled fluid-solid problem efficiently and accurately after several simulations are taken and their results are analyzed.

5:11PM E5.00003 Navier-Stokes adjoint accuracy for aeroacoustic flow control and analysis , RAMANATHAN VISHNAMPET, JONATHAN FREUND, DANIEL BODONY, University of Illinois at Urbana-Champaign — Optimal control based on discrete solutions of the continuous adjoint of the compressible Navier-Stokes equations has been successful for aeroacoustic flows despite discretization truncation errors, which result in an inconsistent sensitivity gradient. For finite resolution simulations, the truncation errors can limit the success of the optimization, especially for turbulent flows; recent evidence of this is presented. The gradient obtained from the discrete adjoint, which is more challenging to formulate but is insensitive to truncation errors, is consistent, and therefore, better suited to minimize our cost functional. We formulate the discrete adjoint of the compressible Navier-Stokes equations using high-order summation-by-parts operators with simultaneous-approximation-term boundary conditions and a high-fidelity time advancement scheme. We show that the continuous and discrete approaches lead to identical adjoint difference equations except near boundaries, at the last time step, and possibly the first few time steps, which affects the gradient accuracy. We evaluate the gradient from the two approaches and discuss the consequences of the errors in the continuous formulation for control optimization in aeroacoustic problems.

5:24PM E5.00004 CFD-based derivative-free optimization using polyharmonic splines, Part 1 , POORIYA BEYHAGHI, DANIELE CAVAGLIERI, THOMAS BEWLEY, Flow Control Lab, UC San Diego — Nonsmooth CFD-based optimization problems are difficult, due both to the nonconvexity of the cost function and to the extreme cost of each function evaluation. In this work, we develop a derivative-free GPS optimization scheme which makes maximum use of each function evaluation. We seek to improve on the efficiency of the existing methods that have been applied to this class of problems (genetic algorithms, SMF, orthoMADS, etc). At each optimization step, the algorithm proposed creates a Delaunay triangulation based on the existing evaluation points. In each simplex so created, the algorithm optimizes a cost function based on a polyharmonic spline interpolant. This interpolation strategy behaves appropriately even when the evaluation points are clustered in particular regions of interest in parameter space (in contrast with the Kriging interpolation strategy used in existing GPS/SMF algorithms). At each optimization step, an appropriately-modeled error function is combined with the interpolant, weighted with a tuning parameter governing the trade-off between local refinement and global exploration.

5:37PM E5.00005 CFD-based derivative-free optimization using polyharmonic splines, Part 2 , DANIELE CAVAGLIERI, POORIYA BEYHAGHI, THOMAS BEWLEY, Flow Control Lab, UC San Diego — The derivative-free optimization algorithm developed in Part 1 of this work (see Beyhaghi et al.).is extended to include (a) a dynamic trade-off between local refinement and global exploration, and (b) to incorporate convex constraints of various types. The resulting algorithm is then verified on representative test functions and compared with competing algorithms. In particular, we will report on recent efforts to develop high-order low-storage IMEX RK schemes for the accurate time integration of the stiff ODEs arising in large-scale DNS and LES simulations.

Sunday, November 18, 2012 4:45PM - 5:50PM –

Session E6 Electrokinetics III 24B - Chair: Ali Mani, Stanford University

4:45PM E6.00001 Electroosmosis in a potassium chloride aqueous solution in a silica nanochannel with counter-charged surface patches[1] , HARVEY ZAMBRANO, MARIE PINTI, A.T. CONLISK, SHAURYA PRAKASH, The Ohio State University, COMPUTATIONAL MICRO- AND NANOFLUIDICS LAB TEAM, MICROSYSTEMS AND NANOSYSTEMS LAB TEAM — Controlling Electroosmotic flow (EOF) in nanochannel is important for several nano and bio-technology applications. In this work, the EOF is studied by conducting Non-Equilibrium MD Simulations (NEMDS) of an electrolyte confined in a silica nanochannel. having dimensions of 34.76 x 2.53 x 7.0 nm. We model a relatively long channel compared to other MD studies in order to investigate in detail the effect of the amorphous walls on the confined aqueous electrolyte. The system was studied as axial electric fields (AEF) were applied and as the surface charge (SC) was modified by implementing counter-charged patches (CP) on the channel walls. From the velocity profiles, a linear response of the system was observed. Smaller velocities were observed for the cases with increasing surface charge on the patches. Our velocities for the reference case with no patches (i.e. bare silica nanochannel) are in agreement with results from previous MD studies. We infer that ionic accumulation on the CP is responsible for the EOF velocity changes for systems with different CP and the same AEF applied. We show that by increasing the SC on a wall, the velocity field decreases monotonically.

[1]This work is supported by Army Research Office (ARO) grant number W911NF1010290.

4:58PM E6.00002 Effect of divalent ions on electroosmotic transport in a sodium chloride aqueous solution confined in an amorphous silica nanochannel[1] , A.T. CONLISK, HARVEY ZAMBRANO, The Ohio State University, NECMETTIN CEVHERI, MINAMI YODA, Georgia Institute of Technology, COMPUTATIONAL MICRO- AND NANOFLUIDICS LAB TEAM, THE FLUIDS, OPTICAL AND INTERFACIAL DIAGNOSTICS LAB TEAM — A critical enabling technology for the next generation of nanoscale devices, such as nanoscale "lab on a chip" systems, is controlling electroosmotic flow (EOF) in nanochannels. In this work, we control EOF in an aqueous sodium chloride (NaCl) solution confined in a silica nanochannel by systematically adding different amounts of divalent ions. Multivalent ions have a different affinity for the silica surface and different hydration characteristics in comparison to monovalent ions. Therefore by adding Mg^{++} and Ca^{++} to the sodium chloride solution, the electroosmotic velocity and the structure of the electrical double layer will be modified. The effects of adding Mg^{++} and Ca^{++} will be compared using non-equilibrium molecular dynamics simulations of the EOF at different electric fields of a NaCl solution in a silica nanochannel with different fractions of Ca^{++} and Mg^{++} ions. In general, the wall zeta-potential magnitude, and hence the EOF velocity, decreases as the Ca^{++} or Mg^{++} concentration increases. The system responds linearly with electric field. We will compare the computational results with the experimental data of Cevheri and Yoda (2012).

[1]This work is supported by Army Research Office (ARO) grant number W911NF1010290.

5:11PM E6.00003 Electrowetting of electrolyte solution in a nanoslit with overlapped electric double layer: continuum approach[1] , IN SEOK KANG, JUNG A. LEE, POSTECH — In a nanotube or nanoslit of O(10nm) lengthscale, the electric double layer (EDL) is expected to be overlapped. For this lengthscale, the continuum approach is still valid. In the present work, electrowetting phenomenon in a nanoslit is analyzed by using the electromechanical method based on the continuum governing equation. From the analysis, we obtain the formula of the extra-pressure that is generated by the electrowetting effect in a nanoslit. We also obtain the deformed shape of the electrolyte-gas interface by using the first order perturbation method. In order to handle the problem analytically, two limiting situations are considered: (i) the low surface potential limit to have a linearized Poisson-Boltzmann equation (PBE), and (ii) the high surface potential limit for which it is assumed that only the counter ions are present inside the nanoslit.

[1]This work has been supported by the BK21 program of the Ministry of Education, Science and Technology of Korea.

5:24PM E6.00004 Nonlinear electrokinetic transport in networks of microscale and nanoscale pores , SHIMA ALIZADEH, MATHIAS B. ANDERSEN, ALI MANI, Department of Mechanical Engineering, Stanford University — The objective of this study is to develop the understanding of nonlinear electrohydrodynamic effects in a wide range of systems including lab-on-a-chip systems, electroosmotic pumps, and, in general, porous media with random or fabricated pore morphology. We present a continuum model in which these systems are described as massive networks of long and thin pores. The thickness of the pores can vary from nanoscale to microscale, corresponding to the highly overlapped electric double layers (EDL) to the thin double layer limit. Within each pore the transport in the wall-normal direction is assumed to be in equilibrium leading to a reduced order model for the axial transport of species in the form of a transient one-dimensional partial differential equation (PDE). PDEs from different pores are coupled through boundary conditions at the pore intersections by proper implementation of the conservation laws. We show that this model can capture important nonlinear dynamics, which are typically ignored in homogenized models. Specifically, our model captures concentration polarization shocks and flow recirculation zones respectively formed when micropores and nanopores are connected in series and in parallel. We present a comparison between our model and recent experiments in microfluidics, and will discuss applications in porous media modeling for energy storage and water purification systems.

5:37PM E6.00005 Large Apparent Electric Size of Solid-State Nanopores Obtained by Focused Ion Beam Milling , REMY FULCRAND, CHOONGYEOP LEE, LAURENT JOLY, ALESSANDRO SIRIA, ANNE LAURE BIANCE, LYDERIC BOCQUET, Laboratoire de Physique de la Matiere Condensee et Nanostructures — Here, we report experimental results that show unexpectedly large ionic conduction in solid-state nano-pores, taking its origin in anomalous entrance effects [1]. The surface conductance inside the nano-pore is shown to perturb the three dimensional electric current streamlines far outside the nano-pore in order to meet charge conservation at the pore entrance. This supports the idea that ion transport is strongly perturbed outside the pore over a healing length given by the so-called Dukin length so as to meet ion current conservation at the entrance of the nano-pore. We developed a simplified analytical model for the conduction in nano-pores, which provides a very good agreement with experimental results. This unexpected effect can be interpreted in terms of apparent electrical size of the nano-pore much larger than its bare geometrical size. Our findings can have a major impact on the electrical detection of translocation events through nano-pores, as well as for ionic transport in biological nano-pores, which use electrical detection of translocation events [2].

[1] C. Lee, L. Joly, A. Siria, A.L. Biance, R. Fulcrand and L. Bocquet, *NanoLetters* **2012**, DOI: 10.1021/nl301412b
[2] L. Song, M.R. Hobaugh, C. Shustak, S. Cheley, H. Goaux *Science* **1996**, 274, 1859.

Sunday, November 18, 2012 4:45PM - 5:50PM –

Session E7 Suspensions I 24C - Chair: Joe Goddard, University of California, San Diego

4:45PM E7.00001 Microstructure and Rheology of Particle Suspension in a Yield Stress Fluid , STEPHANIE DEBOEUF, LUCIE DUCLOUÉ, NICOLAS LENOIR, GUILLAUME OVARLEZ, Laboratoire Navier (Ecole des Ponts ParisTech, IFSTTAR, CNRS UMR8205) — Numerous industrial and natural fluids, such as fresh concrete or debris flows, are made of particles suspended in a yield stress fluid. Such suspensions exhibit complex rheological behaviours: elasto-plasticity, strain hardening, shear-thinning (non constant viscosity), normal stress difference, ... In the light of results about shear-induced microstructure in concentrated suspensions of spheres in Newtonian fluids [Parsi & Gadala-Maria 1987, Brady 1997, Morris 2009, Blanc et al 2011], we experimentally addressed the role of the particle microstructure in this complex rheology. Our model system is made of non-Brownian spherical hard particles suspended in a concentrated emulsion. We associate rheological measurements to microtomography X imaging techniques allowing for high resolution pair distribution functions in three dimensions. We characterize this microstructure for various shear histories (squeeze flow and rotational flow, transient and steady state, various low and high shear rates) and we relate it to their elastic and plastic properties.

4:58PM E7.00002 Effect of sheared-induced diffusion on the transfer of heat across a sheared suspension[1] , BLOEN METZGER, IUSTI CNRS UMR 7343, Marseille, France, XIAOLONG YIN, Petroleum Engineering, Colorado School of Mines, Golden, Colorado, USA, ICSD TEAM, YIN'S GROUP TEAM — Suspensions of non-Brownian particles undergoing shear provide a quasi-unique system where mixing occurs spontaneously at low Reynolds number. In essence, particles behave in the fluid as so many "stirrers." The questions raised are how do they affect the transport of heat/mass across sheared suspensions? What will be the influence of the particle size, their volume fraction and the applied shear? By using an index matched suspension and a laser induced fluorescence imaging technique, we were able to measure individual particle trajectories and correlate the particle diffusion motion to the thermal diffusion of the suspension. Particles cause a significant enhancement ($> 300\%$) of the suspension transport properties. Simulations which combine a Lattice Boltzman technique to solve the flow and a passive Brownian tracer algorithm to solve for the transfer of heat are in very good agreement with experiments.

[1]Funds: ANR JCJC SIMI 9 and IC-Star.

5:11PM E7.00003 Effects of inertia on the steady shear rheology of concentrated emulsions: sign reversal of normal stress differences[1] , PRIYESH SRIVASTAVA, University of Delaware, KAUSIK SARKAR, George Washington University — The shear rheology of moderately concentrated emulsions (5-27% volume fraction) in the presence of inertia is numerically investigated. Typically, an emulsion of viscous drops experiences positive first normal stress difference (N_1) and negative second normal stress difference (N_2), as has also been predicted by perturbative analysis (Choi-Schowalter model) and numerical simulation. However, recently using single drop results we have shown [Li and Sarkar, 2005, J. Rheo, 49, 1377] that introduction of inertia reverses the signs of the normal stress difference in the dilute limit. Here, we numerically investigate the effects of interactions between drops in a concentrated system. The simulation is validated against the dilute results as well as analytical relations. It also shows the reversal of signs for N_1 and N_2 for small Capillary numbers above a critical Reynolds number. The physics is explained by the inertia-induced orientation of the individual drops in shear. Increasing volume fraction increases the critical Reynolds number at which N_1 and N_2change sign. The breakdown of linearity with volume fraction with increasing concentration is also analyzed.

[1]Partially supported by NSF.

5:24PM E7.00004 Buckling of particle-laden interfaces , THEO KASSUGA, JONATHAN ROTHSTEIN, University of Massachusetts Amherst — Particle-laden interfaces have been shown to have very interesting physical behavior, such as being able to resist compressive and shear stresses, and helping stabilize emulsions and foams. In this work, we study the buckling of an oil-water interface populated by micron-sized latex particles using a Langmuir trough. We extend pre-existing results to the micron-sized range with different density ratio and show that the existing theoretical framework still applies as a prediction of the dominant wrinkle wavelength. However, histograms show that the wavelength distribution has two peaks, which indicates that there occurs a cascading phenomenon similar to that observed in thin solid sheets. We can characterize this by tracking the position within the particle raft where cascading occurs, the wavelength of the resulting wrinkles, and their width along the crest.

5:37PM E7.00005 Suspensions with a tunable effective viscosity[1] , PHILIPPE PEYLA, SALIMA RAFAI, Grenoble University, France, LEVAN JIBUTI, Bayreuth University, Germany — In this work, we conduct a numerical investigation on sheared suspensions of non-colloidal spherical particles on which a torque is applied. Particles are mono-dispersed and neutrally buoyant. Since the torque modifies particles rotation, we show that it can indeed strongly change the effective viscosity of semi-dilute or even more concentrated suspensions. We performed our calculations up to a volume fraction of 0.28. And we compare our results to data obtained at 0.40 by other authors with a totally different numerical method. Depending on the torque orientation, one can increase (decrease) the rotation of the particles. This results in a strong enhancement (reduction) of the effective shear-viscosity of the suspension. We construct a dimensionless number Q which represents the average relative angular velocity of the particles divided by the vorticity of the fluid generated by the shear flow. We show that the contribution of the particles to the effective viscosity can be suppressed for a given and unique value of Q independently of the volume fraction. In addition, we obtain a universal behavior (independent of the volume fraction) when we plot the relative effective viscosity divided by the relative effective viscosity without torque as a function of Q. Finally, we show that a modified Faxen law can be equivalently established for large concentration.

[1]We thank the ANR MOSICOB and MICMACSWIM.

Sunday, November 18, 2012 4:45PM - 5:50PM –

Session E8 Particles: Collisions 25A - Chair: Jane Wang, Cornell University

4:45PM E8.00001 Computing Particle Collisions in Fluids by Incorporating the Lubrication Theory in the Immersed Interface Method , ACMAE EL YACOUBI, Cornell University, SHENG XU, Southern Methodist University, Z. JANE WANG, Cornell University — The interactions of particles in fluids are key to understanding collective behavior of particle suspensions. To compute the dynamics of these systems in the high particle-density limit, one has to treat the collision of particles. There has been experimental and theoretical studies to understand the dynamics of particle collisions in fluids. However, direct numerical simulation remains a challenge. The small gap introduces difficulties in spatial resolution, as it would require successive local refinements of the grid. A scheme with a fixed grid resolution would break down when the gap falls below a threshold. There have been various $ad\ hoc$ methods using a repulsive force or modified dry collision equations. However, they can lead to unrealistic dynamics such as the rebound of particles. In this talk, we will present a computational method which applies the lubrication theory in the interstitial region between particles. We will describe the numerical implementation in the immersed interface method framework. The fluid velocity and pressure gradient in the gap are solved for analytically, and are used in the expression of the singular forces. We test our computational scheme by checking against analytical solutions of interactions between a falling cylinder and a wall.

4:58PM E8.00002 The Importance of Collisions in the Simulation of Lunar Soil Ejection during Spacecraft Landing , KYLE BERGER, University of Colorado Boulder, PHILIP METZGER, NASA Kennedy Space Center, CHRISTINE HRENYA, University of Colorado Boulder — When a spacecraft lands on the Moon, the rocket exhaust causes lunar soil to be ejected. Due to the lack of atmospheric drag and reduced gravity, the ejected soil can be extremely hazardous to equipment and/or persons both close and far from the landing point. Current models for the ejection are based on single-particle trajectories. Here we critically assess the impact of collisions on erosion. Specifically, the discrete element method (DEM), which incorporates collisions directly, is used. The system examined is located 6m from the impingement point of the rocket and includes the lift and drag forces from the exhaust plume, as well as lunar gravity. A one-way coupling is utilized to describe how the plume affects the particles. Two versions of the DEM are used: one which resolves collisions using a soft-sphere model and another which ignores the collisions (and is thus similar to the single-particle trajectory calculations). In addition, both non-dissipative and dissipative collisions are considered in the collisional model. Somewhat surprisingly, the erosion rate of the collision-less case lies between that of the dissipative and non-dissipative collisional cases. In addition, a sensitivity analysis to collisional input parameters is also performed.

5:11PM E8.00003 Rheology of colloidal suspensions measured by dragging a probe , XIN DU, Graduate Student, Emory University, PIOTR HABAS, Associate Professor, Saint Joseph's University, ROSEANNA ZIA, Post-doc, Princeton University, ERIC WEEKS, Professor, Emory University — We use active microrheology to study the rheological properties of colloidal suspensions at moderate volume fractions. Traditionally, the rheology of complex fluids is experimentally studied using macroscopic mechanical rheometers. Alternatively, single-particle tracking—microrheology—can be utilized to measure material properties. Microrheology involves the tracking the motion of a probe particle embedded in a complex fluid. In passive microrheology, the motion of the probe particle is driven by thermal fluctuations. Here we study non-equilibrium systems via active microrheology, in which a magnetic probe particle is dragged by a constant external magnetic force through a suspension of colloidal particles. By tracking the mean and mean-square probe motion, the viscosity, diffusivity, and normal stresses are obtained. The effective viscosity of the suspension is determined from the mean velocity of the probe particle. The velocity fluctuations of the probe which are parallel and perpendicular to the mean velocity direction produce force-induced probe diffusion, are measured by the mean-square displacement of the probe. By applying recent theory, the two measurements are combined to understand other rheological properties of the complex fluid such as normal stresses. Our results are in good agreement with macroscopic rheology of similar suspensions, demonstrating that the microscopic technique may be useful for cases when only small sample quantities are available.

5:24PM E8.00004 Stability of bed particles near the critical threshold of motion , JULIAN SIMEONOV, JOSEPH CALANTONI, Marine Geosciences Division, Naval Research Laboratory, Code 7434, Stennis Space Center, MS 39529, USA — The unsteady flow above a rough bed of mobile spherical particles is investigated with Direct Numerical Simulations. The velocity and pressure are resolved at sub-particle scales using a Cartesian grid numerical method based on a discontinuous extension of the pressure Poisson equation across particle boundaries. The hydrodynamics is fully resolved everywhere except in the gap between colliding particles when the latter becomes smaller than the grid step. To correctly predict momentum dissipation due to the viscous flow in the unresolved gap between colliding particles, we add analytical lubrication forces to the numerically resolved hydrodynamic force. The normal and tangential forces due to mechanical contact are modeled using a linear elastic-plastic law (soft-sphere) and a history dependent friction law, respectively. The collision model is validated against experimental data for normal and oblique immersed collisions of spherical particles. The lubrication effects during weak collisions are essential for damping out the flow-induced vibrations of bed particles confined in surface pockets. The results from our numerical simulations for the initiation of motion are compared with existing laboratory data.

5:37PM E8.00005 Observation of the Sling Effect , GREGORY BEWLEY, EWE WEI SAW, EBERHARD BODENSCHATZ, Max Planck Institute (DS) Goettingen, INTERNATIONAL COLLABORATION FOR TURBULENCE RESEARCH COLLABORATION — We report the first experimental observations of the sling effect, by which fluid turbulence increases the rate of collisions between suspended droplets. We put liquid water-alcohol droplets in a turbulent air flow, and followed their motions in three dimensions with two cameras. The turbulence was approximately isotropic with a Taylor Reynolds number of about 200. The resulting droplet Stokes numbers were between 0.1 and 0.6, depending on the intensity of the turbulence and the size of the droplets. We used two-droplet statistics to characterize the droplet velocity field and its gradient. The droplet velocity field contained gradients that were large enough relative to the droplet response time for slings to occur, according to the theory. The large negative gradients steepened just as was predicted for slings. During this steepening, the droplet gradients decoupled from the background fluid gradients.

Sunday, November 18, 2012 4:45PM - 5:37PM –

Session E9 Aerodynamics I 25B - Chair: Kamran Mohseni, University of Florida

4:45PM E9.00001 Aerodynamic Improvements to Cargo Carrying Rail Cars due to Roof Modifications , ROBERT CONDIE, DANIEL MAYNES, Brigham Young University — The aerodynamic drag associated with the transport of commodities by rail is becoming increasingly important as the cost of diesel fuel increases. We provide an assessment of the influence of the roof structure on aerodynamic performance of two dissimilar rail cars, namely automobile carrying cars and coal carrying cars. Currently, the roof material for automobile carrying rail cars is corrugated steel, with the corrugation aligned perpendicular to the direction of travel. Coal cars are currently left uncovered for loading convenience and on the return leg from the power plant are empty. Aerodynamic drag data have been obtained through wind tunnel testing on 1/29 scale models to understand the savings that may be realized by judicious modification to the tops of both these car types. For the automobile-carrying cars, testing is performed for the corrugated and smooth roof configurations. This modification alone has the potential of reducing the car drag coefficient by nominally 25%. A broader study is performed for the coal cars, with data being acquired for coal filled models, empty models, and several cover prototype configurations. The results reveal that implementation of a cover may yield reductions in the aerodynamic drag for both coal filled (nominally 7%) and empty coal cars (nominally 30%).

4:58PM E9.00002 Flow over a Ram-Air Parachute Canopy[1] , ALI ESLAMBOLCHI, HAMID JOHARI, California State University, Northridge — The flow field over a full-scale, ram-air personnel parachute canopy was investigated numerically using a finite-volume flow solver coupled with the Spalart-Allmaras turbulence model. Ram-air parachute canopies resemble wings with arc-anhedral, surface protuberances, and an open leading edge for inflation. The rectangular planform canopy had an aspect ratio of 2.2 and was assumed to be rigid and impermeable. The chord-based Reynolds number was 3.2 million. Results indicate that the oncoming flow barely penetrates the canopy opening, and creates a large separation bubble below the lower lip of canopy. A thick boundary layer exists over the entire lower surface of the canopy. The flow over the upper surface of the canopy remains attached for an extended fraction of the chord. Lift increases linearly with angle of attack up to about 12 degrees. To assess the capability of lifting-line theory in predicting the forces on the canopy, the lift and drag data from a two-dimensional simulation of the canopy profile were extended using finite-wing expressions and compared with the forces from the present simulations. The finite-wing predicted lift and drag trends compare poorly against the full-span simulation, and the maximum lift-to-drag ratio is over-predicted by 36%.

[1]Sponsored by the US Army NRDEC

5:11PM E9.00003 Flow in the near wake of hemispherical parachute shapes , JEFFREY YOUNG, MARIA-ISABEL CARNASCIALI, Univestity of New Haven, MIKE KANDIS, Pioneer Aerospace — A CFD study was conducted using ANSYS to investigate the pitch-stability of several hemispherical parachute geometries at varying Reynolds numbers. In actuality, the parachute itself is not a rigid body and large variations in the parachute geometry can occur due to the flexibility of the parachute fabric. This factor combined with flow through gaps/open areas provide for a much more complex wake than that of a simple bluff body like a disc or sphere. In some cases, Vortex Shedding or alternating vortices are generated which cause oscillations in the axial (i.e., drag force) and normal (i.e., lift force) forces that lead to pitching/oscillations. This study investigated the flow in the near wake of hemispherical parachute shapes (assumed to be rigid) having various sized gaps/open areas positioned at distinct locations to determine which designs resulted in "less severe" Vortex Shedding. The design features (i.e., size and location of the gaps) that provided the smallest variation/fluctuation in the normal forces were identified and compared to actual parachute designs.

5:24PM E9.00004 A lifting surface approximation for roll stall of Micro Aerial Vehicles , MATT SHIELDS, KAMRAN MOHSENI, University of Florida — The lateral stability of Micro Aerial Vehicles (MAVs) has been known to be adversely affected by the low aspect ratio (LAR) nature of these aircraft. While this has typically been attributed to the small moments of inertia about the plane of symmetry, recent experimental results display the development of a significant roll stability derivative ($C_{l,\beta}$) for flat plate (0% camber) wings. The roll moment can be attributed to the asymmetric development of the tip vortices of a yawed wing and the resulting deviation from the wing loading at zero sideslip. Furthermore, results indicate that a harmonic yaw oscillation at increasing angular velocities results in a delay effect as the formation of the tip vortex is affected by the rotation of the wing; that is, the roll moment does not reach its steady value at a given yaw angle until after the model yaws past the angle. A model based on modified lifting surface theory is developed to determine the influence of the induced velocities of the skewed tip vortices on the lateral loading of both the static and oscillating wing; experimentally determined parameters are used to compensate for the separated flow experienced by MAV wings and not considered in conventional lifting surface methods.

Sunday, November 18, 2012 4:45PM - 5:50PM –

Session E10 Instability: Jets, Wakes and Shear Layers III: Wakes II 25C - Chair: Sherwin Maslowe, McGill University

4:45PM E10.00001 Hub vortex helical instability as the origin of wake meandering in the lee of a model wind-turbine , FRANCESCO VIOLA, LFMI-EPFL Lausanne, GIACOMO VALERIO IUNGO, WIRE-EPFL Lausanne, SIMONE CAMARRI, DIA - Università di Pisa, FERNANDO PORTE-AGEL, WIRE-EPFL Lausanne, FRANCOIS GALLAIRE, LFMI-EPFL Lausanne — Wind tunnel measurements were performed for the wake produced by a three-bladed wind turbine immersed in uniform flow. These tests show the presence of a vorticity structure in the near wake region mainly oriented along the streamwise direction, which is denoted as hub vortex. The hub vortex is characterized by oscillations with frequencies lower than the one connected to the rotational velocity of the rotor, which are ascribed to wake meandering by previous works. This phenomenon consists in transversal oscillations of the wind turbine wake, which are excited by the shedding of vorticity structures from the rotor disc acting as a bluff body. In this work temporal and spatial linear stability analyses of a wind turbine wake are performed on a base flow obtained through time-averaged wind tunnel velocity measurements. This study shows that the low frequency spectral component detected experimentally is the result of a convective instability of the hub vortex, which is characterized by a counter-winding single-helix structure. Simultaneous hot-wire measurements confirm the presence of a helicoidal unstable mode of the hub vortex with a streamwise wavenumber roughly equal to the one predicted from the linear instability analysis.

4:58PM E10.00002 Inviscid Instability of a Trailing Vortex , SHERWIN MASLOWE, JAN FEYS, McGill University — A similarity solution for an aircraft trailing vortex valid far downstream of its wingtip was found by Batchelor (1964). Its linear stability has been the focus of many papers following the pioneering work of Lessen et al. (1974). In the parallel flow version of Batchelor's solution, the azimuthal and axial velocities can be written $W = e^{-r^2}$ and $V = q(1 - e^{-r^2})/r$. The swirl component $V(r)$ is stabilizing and for $q > 2.31$ the vortex is stable. It was pointed out, however, by Spalart (1998) that closer to the aircraft a family of profiles found by Moore & Saffman (1973) provide a more accurate description of the axial flow. Comparing with the recent experiments of Lee & Pereira (2010), we find that the latter profiles better describe the axial flow deficit in the vortex core. We present results for the Moore & Saffman profiles showing them to be considerably more unstable than the Batchelor vortex.

5:11PM E10.00003 Reflectional symmetry breaking of the separated flow over 3D bluff bodies[1] , MATHIEU GRANDEMANGE, ENSTA ParisTech / PSA Peugeot Citroën, MARC GOHLKE, PSA Peugeot Citroën , OLIVIER CADOT, ENSTA ParisTech — The first experimental observation of a permanent reflectional symmetry breaking (RSB) is reported for a laminar three-dimensional wake. Based on flow visualizations, a first bifurcation from the trivial steady symmetric state to a steady RSB state is characterized at Re = 340. The RSB state becomes unsteady after a second bifurcation at Re = 410. It is found that this RSB regime is persistent at large Reynolds numbers and responsible for a bi-stable turbulent wake.

[1]The authors are indebted to R. Godoy Diana, V. Raspa and B. Thiria from ESPCI (Paris, France) for lending their low speed hydrodynamics tunnel.

5:24PM E10.00004 ABSTRACT WITHDRAWN –

5:37PM E10.00005 Oscillation, bifurcation and growth of modal instability in bluff-body wakes: a new understanding[1] , AMALENDU SAU, Gyeongsang National University — Past experimental findings suggest during nucleation of a vortex over a bluff body the near-wake instability initiates a wavy trail even at *Re*=30, and wavelength of this oscillation decreases with increasing *Re* while amplitude increases. We discovered that, such a wavy oscillation has a strong spanwise counterpart which gets fast augmented with *Re*, and enforces growth of a new class of bifurcations along the Kármán vortex cores. Notably, the detected pressure oscillation along a vortex core reaches a threshold value at the onset of shedding process and initiates growth of a Hopf bifurcation in spanwise coordinate. With *Re* the pressure oscillation gains momentum; enforcing occurrence of multiple local pressure maxima and bifurcations along a vortex coreline. Our detailed simulations with square cylinders of different aspect ratios and *Re* up to 240 unfold development of two physically distinct stages of spanwise wake undulations and bifurcations. While growth of uniform length-scaled bifurcations and their spatio-temporal swapping initiate formation of "Mode A" instability structures, a transition scenario for the "Mode B" is prompted with the eruption of variable length-scaled spanwise bifurcations.

[1]Travel grant from DRC, GNU is gratefully acknowledged.

Sunday, November 18, 2012 4:45PM - 6:03PM –

Session E11 Rotating Flows I: Rotating Convection 26A - Chair: Antonio Rubio, University of Colorado

4:45PM E11.00001 Generation of a large-scale barotropic circulation in rotating convection[1]

, ANTONIO RUBIO[2], KEITH JULIEN[3], JEFFREY WEISS[4], University of Colorado, Boulder — We recently reported on the existence of a slow-growing large scale barotropic mode in DNS of rotating Rayleigh-Benard convection using the non-hydrostatic balanced geostrophic equations (NHBGE) (Julien et al 2012). Such large scale modes had been previously observed as an inverse cascade in stable layer quasi-geostophic dynamics or via instability mechanisms of thermal Rossby waves occuring in presence of sloping endwalls (i.e quasi-geostrophic beta-convection). In this talk we report on the early time history of this large scale mode and discuss the generating physical mechanism as a "symmetry-breaking" forcing function of the barotropic vorticity equation. Impacts of the large scale barotropic mode on the smaller scale baroclinic components of the flow are detailed with a specific emphasis on the changing nature of the heat transport as the barotropic mode evolves.

[1]This work was supported by the National Science Foundation under FRG grants DMS-0855010 and DMS-0854841. Computational resources supporting this work were provided by the NASA High-End Computing (HEC) Program through the NASA Advanced Supercomputing (NAS)

[2]Department of Applied Mathematics

[3]Department of Applied Mathematics

[4]Department of Atmospheric and Oceanic Sciences

4:58PM E11.00002 Acceleration PDFs of particles in rotating turbulent convection[1]

, HERMAN CLERCX, PRASAD PERLEKAR, VALENTINA LAVEZZO, FEDERICO TOSCHI, Physics Department, Eindhoven University of Technology — Particle dispersion in buoyancy-driven rotating turbulent flows has direct relevance for many industrial and environmental applications. We have used a Lattice Boltzmann Method coupled with Lagrangian particle tracking algorithm to investigate the behaviour of passive and inertial particles released in turbulent rotating Rayleigh-Bénard (RB) convection. The flow domain is horizontally periodic and vertically confined. Both the gravity and the rotation vector are oriented in the vertical direction. Here we present the results of the acceleration PDFs of particles in both non-rotating and strongly rotating RB convection. It is found that the bulk acceleration PDF in non-rotating RB turbulence is like in homogeneous isotropic turbulence whereas rotation introduces anisotropy similar to acceleration PDFs obtained from experiments in (isothermal) forced rotating turbulence [1]. These results and those obtained for inertial particles will be discussed.

[1] L. Del Castello and H.J.H. Clercx, Phys. Rev. Lett. **107**, 214502 (2011).

[1]PP and VL were financially supported by the Foundation for Fundamental Research on Matter (FOM), which is part of NWO. This work was sponsored by NWO-NCF (SH-176).

5:11PM E11.00003 Low rossby number heat transport in rotating Rayleigh-Benard convection[1]

, KEITH JULIEN, ANTONIO RUBIO, University of Colorado at Boulder, GEOFFREY VASIL, CITA, University of Toronto, EDGAR KNOBLOCH, University of California at Berkeley — Recent laboratory experiments of turbulent rotating Rayleigh-Bénard convection, performed entirely within the regime of strong rotational constraint, have revealed a sharp transition in the scaling of the heat transport as a function of the thermal forcing. This is embodied by the nondimensional Nusselt-Rayleigh scaling law, $Nu \propto Ra^{\alpha}$, where a steep scaling regime ($\alpha > 1$) gives way to a comparatively shallower regime ($\alpha < 1/2$) typical of non-rotating turbulent convection. A crossover between the thermal and viscous boundary layers has been proposed as the root-cause of this remarkable result, yet a similar transition is found in the presence of stress-free boundary conditions where viscous layer boundary layers are absent. Unfortunately, the dynamics within the thermal boundary layer remain poorly understood due to resolution challenges at low Rossby number. Utilizing numerical simulations of the asymptotically exact nonhydrostatic balanced geostrophic equations we present an alternative explanation, not reliant on the form of the mechanical boundary conditions, but based on loss of geostrophic balance within the thermal boundary layers as a result of vigorous vortical motions. We show that the bottleneck for heat transport is the turbulent interior.

[1]NSF FRG DMS-0855010 and DMS-0854841

5:24PM E11.00004 Heat Transport and Local Temperature Measurements of Geostrophic Rotating Thermal Convection

, ROBERT ECKE, SCOTT BACKHAUS, SRIDHAR BALASUBRAMANIAN, Los Alamos National Laboratory — Rotating Rayleigh-Benard convection is an idealized model of geophysical convective motions where buoyancy and rotation compete. The parameters governing such flows are the Rayleigh number Ra proportional to ΔT across the cell height h, the Taylor number Ta proportional to Ω^2 where Ω is the angular rotation rate, and the Prandtl number Pr. In the turbulent state, experiments have demonstrated that normalized heat transport Nu for the rotating state at small $Ro = \sqrt{Ra/(PrTa)}$ scales in the same manner as the non-rotating heat transport with a small enhancement of heat transport that depends on Ra and Pr. We explore global heat transport and local temperature measured at multiple vertical positions along the cell center line of a square convection cell with aspect ratio $\Gamma = L/h \approx 4$ where L is a lateral side and $h = 12.1$ cm is the cell height. We focus on the Ra range $5 \times 10^6 < Ra < 5 \times 10^8$ for $5 \times 10^8 < Ta < 5 \times 10^{10}$ from onset up to the crossover to turbulent scaling where $Nu \sim Ra^{0.29}$. We report on the scaling of Nu with Ra at constant Ta in that range and infer local convective structure from vertical spatial correlation of temperature fluctuations.

5:37PM E11.00005 Flow structure in turbulent rotating Rayleigh–Bénard convection

, RUDIE KUNNEN, YOANN CORRE[1], HERMAN CLERCX, Eindhoven University of Technology — Turbulent Rayleigh-Bénard convection is usually studied in an upright cylinder. The addition of axial rotation has profound effects on the flow structuring. The well-known large-scale circulation (LSC) of the non-rotating case is still found at low rotation rates but is replaced by an irregular array of vertically aligned vortical plumes at higher rotation rates. We report PIV measurements of turbulent rotating convection in a cylindrical cell of diameter-to-height aspect ratio $\Gamma = 1/2$ at Rayleigh number $Ra = 4.5 \times 10^9$ and at many rotation rates covering both the LSC and the vortical-plume regime. We focus on: (i) the azimuthal precession of the LSC, (ii) collective motions of the vortical plumes, and (iii) the sidewall boundary layers. With these results we can clarify remarkable differences between the $\Gamma = 1$ and $\Gamma = 1/2$ cases reported recently in the literature.

[1]Traineeship project carried out in Eindhoven as part of Master's Degree at Université Paris–Sud, France.

5:50PM E11.00006 DNS of turbulent co- and counterrotating Taylor Couette flow up to Re=30,000

, BRUNO ECKHARDT, HANNES BRAUCKMANN, Philipps-Universitaet Marburg — We study global and local torque fluctuations in turbulent Taylor-Couette flows for shear Reynolds numbers Re_S up to 3×10^4 at various mean rotations for radius ratios $\eta = 0.71$ and 0.5 and $\Gamma = 2$. Convergence of simulations is tested using three criteria of which the agreement of dissipation values estimated from the torque and from the volume dissipation rate turns out to be most demanding. The typical spatial distribution of the different convective and viscous contributions to the local current are identified and PDF's of local current fluctuations calculated. The results agree with experimental observations after an additional spatial average to account for finite resolution. Simulations realising the same shear $Re_S \geq 2 \times 10^4$ show a maximum in torque for moderate counter-rotation. For lower values $Re_S \leq 4 \times 10^3$ the torque features a maximum for a stationary outer cylinder. For stronger counter rotation the flow develop intermittently fluctuating boundary layers near the outer cylinder. We demonstrate the phenomenon in direct numerical simulations and propose a theoretical model for the critical value in the rotation ratio that agrees well with the observations.

Sunday, November 18, 2012 4:45PM - 5:37PM –

Session E12 Vortex III 26B - Chair: Enrico Fonda, University of Maryland

4:45PM E12.00001 Vortex Motion in Superfluid 4He: Reformulation in the Extrinsic Vortex Filament Coordinate Space , BHIMSEN SHIVAMOGGI, University of Central Florida — Vortex motion in superfluid ^{4}He is considered by reformulating the Hall-Vinen equation (originally formulated in the intrinsic geometric parameter space) in the extrinsic vortex filament coordinate space which is shown to provide a useful alternative approach [1]. It provides insights into the physical implications of one aspect of the friction term, associated with the friction coefficient α, in the Hall-Vinen notation, as well as the previous neglect of the other aspect, associated with the friction coefficient α'. A nonlinear Schrodinger equation for the propagation of nonlinear Kelvin waves on a vortex filament in a superfluid is given. The localized vortex kink solution of this equation is shown to be driven unstable by the normal fluid flow along the vortex in qualitative similarity with the Donnelly-Glaberson instability of Kelvin waves on a vortex. Though the friction term associated with α, for very small α, has little capacity to make significant contribution to the vortex motion in a quantitative way, it appears to be able to influence the vortex kink characteristics in a qualitative way.

[1] B.K. Shivamoggi: *Phys. Rev. B* **84**, 012506, (2011).

4:58PM E12.00002 Visualizing helium II counterflow turbulence around a cylinder[1] , TYMOFIY CHAGOVETS, STEVEN VAN SCIVER, National High Magnetic Field Laboratory — We report an experimental investigation of counterflow He II around a cylinder, thermally induced through vertical channels using an optical method of flow visualization. Previous studies using particle image velocimetry (PIV) and polymer microspheres as tracers demonstrated large scale turbulent eddies both upstream (as measured by the normal fluid flow direction) and downstream of the cylinder. An interpretation of this effect requires more detailed analysis of interaction of particles with normal fluid component and quantized vortex lines. Our recent experiments using solid hydrogen particles and the particle tracking technique (PTV) will be analyzed and results will be compared with the previous experiments. In this context, recent progress in our study of the observed large scale vortex structures in helium II counterflow will be discussed.

[1]Work supported by the US Department of Energy.

5:11PM E12.00003 Frozen particles as cryogenic fluid tracers: observation of Kelvin waves in superfluid helium , ENRICO FONDA, University of Maryland, College Park - New York University, DAVID P. MEICHLE, University of Maryland, College Park, NICHOLAS T. OUELLETTE, Yale University, KATEPALLI R. SREENIVASAN, New York University, DANIEL P. LATHROP, University of Maryland, College Park — Injecting a dilute mixture of a seed gas into a cryogenic fluid produces a mist of frozen particles. These particles can be effective tracers for studying both classical and quantum cryogenic fluids. The particles resulting from injecting a hydrogen mixture in liquid helium right above the lambda transition, subsequently allowing the fluid to cool down below that temperature, allow visualization of thermal counterflow, quantized vortices and vortex reconnection. This technique is unsuitable for detailed study of vortex dynamics below 2K. We have recently discovered a new method for producing atmospheric sub-micron particles directly into superfluid helium, allowing us to study vortices at temperatures between 1.8K and 2K. By visualizing the reconnection of long vortices populating the system in that temperature regime, we have made the first direct observations of Kelvin waves in superfluid helium and characterized a prototypical event.

5:24PM E12.00004 Experimental characterization of Kelvin waves on quantized vortices following reconnection , DAVID P. MEICHLE, University of Maryland, College Park, ENRICO FONDA, New York University - University of Maryland, College Park, NICHOLAS T. OUELLETTE, Yale University, DANIEL P. LATHROP, University of Maryland, College Park — The superfluid state of He4 exhibits reconnection of quantized vortices. Flow visualization is made possible by injecting a dilute mixture of seed gases (or atmosphere) which freeze into sub-micron tracer particles and decorate the vortex lines. Using this technique, we have for the first time directly observed the excitation of a self-similar traveling helical perturbation to a vortex core (Kelvin wave) following a reconnection of two vortices. Detailed analysis and comparison to several semi-classical theories of vortex waves will be presented, along with other recent visual observations in superfluid He flows.

Sunday, November 18, 2012 4:45PM - 5:37PM –

Session E13 Geophysical: General III 27A - Chair: James Rottman, University of California, San Diego

4:45PM E13.00001 Trapped subsurface oil plumes and critical escape phenomena[1] , CHUNG-NAN TZOU, ROBERTO CAMASSA, ZHI LIN, RICH MCLAUGHLIN, KEITH MERTENS, BRIAN WHITE, University of North Carolina — A critical phenomenon concerning the escape/trap of buoyant miscible plumes rising through strongly stratified fluids is presented experimentally and theoretically. The criticality is determined by the distance between plume release height and depth of ambient density transition. For fluid released closer to the background density transition than this critical distance, the buoyant fluid escapes and rises indefinitely. For fluid released further than this critical distance, the buoyant fluid is forever trapped within the fluid. Two new mathematically exact formulas will be presented for the cases of linear and sharp ambient stratification and they show quantitative agreement with experiments. The new solution for linear stratification is analyzed in the limit of vanishing transition layer thickness. The analytic solution for sharp stratification is shown to accurately estimate the depth at which subsurface plumes trapped during the Deepwater Horizon oil disaster. Also, a dimensional analysis argument is presented which captures the essential physics to provide a simple understanding of this phenomenon.

[1]We gratefully acknowledge support from NSF CMG ARC-1025523, NSF RAPID CBET-1045653, NSF DMS-1009750 and NSF RTG DMS-0943851.

4:58PM E13.00002 Numerical simulation of jets generated by a sphere moving vertically in stratified fluids , HIDESHI HANAZAKI, SHOTA NAKAMURA, Kyoto University — Numerical studies are performed on the flow past a sphere moving vertically at constant speeds in a stratified fluid. Initial unsteady development of the flow shows that violation of density conservation due to diffusive effects on the sphere surface is the key process for the generation of jets observed in the experiments, since the fluid which was originally at dragged downward would indefinitely long distance unless the isopycnal surface is not teared off. Initially, when the diffusion is negligible, the density is conserved so that the density contours are simply pulled down by the sphere. As the diffusion becomes larger, fluid of unconserved density generates a jet. The violation of density conservation on the sphere surface occurs first near the equator of the sphere where the diffusion is large, and the diffusion becomes larger poleward to finally become effective at the rear/upper stagnation point.

5:11PM E13.00003 On the validity of Kraichnan scalings for forced two-dimensional turbulence[1] , JEROME FONTANE, Universite de Toulouse, ISAE, DAEP, DAVID DRITSCHEL, RICHARD SCOTT, School of Mathematics and Statistics, University of St Andrews — We examine the validity of the scaling laws derived by Kraichnan (1967) for forced two-dimensional turbulence. We use a new numerical technique (Dritschel & Fontane 2010) to reach higher Reynolds number than previously accessible with classical pseudo-spectral methods. No large scale friction or hypo-diffusion is used in order to avoid any distortion of the inverse cascade and to be in agreement with the theoretical framework of Kraichnan. Both spectral and spatial forcing are considered and statistical convergence is obtained through large simulation ensembles. A steeper energy spectrum proportional to k^{-2} is observed for the inverse cascade in place of the classical $k^{-5/3}$ prediction. This steepening is shown to be associated with a faster growth of energy at large scales, scaling like t^{-1} rather than Kraichnan's prediction of $t^{-3/2}$. The deviation from Kraichnan's theory is related to the emergence of a vortex population dominating the distribution of energy across scales, and whose number density and vorticity distribution with respect to vortex area are related to the shape of the enstrophy spectrum.

[1]Funded by European Community in the framework of the CONVECT project under Grant number PIEF-GA-2008-221003.

5:24PM E13.00004 Perturbations from 2D Navier-Stokes: rapidly rotating and weakly stratified Boussinesq flow[1] , JARED WHITEHEAD, BETH WINGATE, Los Alamos National Laboratory — Relying on the derivation of "slow" equations in the limit of rapid rotation (Rossby number 0) and weak stratification (Froude number 1), we demonstrate that the 3D Boussinesq equations (for any Rossby and Froude numbers) can be written as 2D Navier-Stokes plus a Reynolds stress, and two passive advection equations (with additional active Reynolds stresses). The impact this formulation has on the exchange of horizontal and vertical kinetic energy with potential energy is discussed, as well as the breakdown of the potential enstrophy into components relative to a passive density stratification. These results are elucidated with direct numerical simulations that consider rapid rotation (small but finite Rossby number) with weak stratification.

[1]We gratefully acknowledge the support of the Department of Energy through the LANL/LDRD program.

Sunday, November 18, 2012 4:45PM - 5:50PM –

Session E14 Experiments: Turbulence 27B - Chair: John Eaton, Stanford University

4:45PM E14.00001 Transition to Turbulence in Oscillatory Flow for Pulse Tube Cryocoolers , MEGHAN MCNULTY, Department of Mechanical Engineering, Virginia Tech, Blacksburg, VA, BENJAMIN JEWELL, THOMAS FRASER, Air Force Research Laboratory / RVSS, Kirtland AFB, NM — Pure (zero-mean) oscillatory flows are less studied than the pulsatile flows found in biology, yet they are frequently used in low-temperature cooling systems, like the pulse tube cryocooler (PTC). PTCs have high potential for extended lifespans and reductions in size, weight, and power compared to other cryocoolers, but advancements of the technology have been hampered by the lack of knowledge of the working fluid's behavior. While design guides assume laminar flow in the pulse tube, an evaluation of PTCs in literature using the Womersley parameter $\alpha = a(\omega/\nu)^{1/2}$ and the oscillatory Reynolds number $Re_\delta = U(2/\omega\nu)^{1/2}$ indicates the probability of transitional or turbulent flow. Because PTC operation relies on thermal stratification of the oscillating internal gas, turbulent mixing will significantly reduce performance. We quantify the fluid flow within a PTC under near-operational conditions using planar particle image velocimetry (PIV) and calculate the first full-field velocity measurements that provide insight to the presence of transitional or turbulent flow and the physics that underlie experimentally observed "streaming effects."

4:58PM E14.00002 Time-Resolved, Two-Dimensional Imaging of Scalar Mixing in Turbulent Gas-Phase Jets , MICHAEL PAPAGEORGE, JEFFREY SUTTON, Ohio State University — The objective of this work is to examine the dynamics of scalar mixing in turbulent, gas-phase jets using kHz-rate laser diagnostics. The research is underpinned by a new High Energy Pulse Burst Laser System (HEPBLS), which is capable of delivering more than 150 high-energy ($> 500mJ$) pulses with repetition rates exceeding 10 kHz. The unique system allows for the extension of traditionally low repetition-rate planar laser techniques such as Rayleigh scattering and Planar Laser-Induced Fluorescence (PLIF) to high-speed imaging applications. In this study, two turbulent jets with Reynolds number equal to $10,000$ and $15,000$ (based on jet diameter) are used to study time-dependent scalar mixing and dissipation processes. Temporally-resolved, two-dimensional images of the mixture fraction and scalar dissipation rate fields are obtained at axial positions of $\frac{x}{D} = 10$ to $\frac{x}{D} = 40$, revealing the highly transient mixing topology within turbulent jets. Averaged results are validated against similar imaging techniques at low repetition rates and known turbulent scaling laws. In addition to "real time" visualization, the scalar mixing dynamics are characterized with temporal and spatial statistics.

5:11PM E14.00003 On investigating wall shear stress in two-dimensional plane turbulent wall jets , FARAZ MEHDI, University of New Hampshire, GUNNAR JOHANSSON, Chalmers University of Technology, CHRISTOPHER WHITE, University of New Hampshire, JONATHAN NAUGHTON, University of Wyoming — Mehdi & White [Exp Fluids 50:43–51(2011)] presented a full momentum integral based method for determining wall shear stress in zero pressure gradient turbulent boundary layers. They utilized the boundary conditions at the wall and at the outer edge of the boundary layer. A more generalized expression is presented here that uses just one boundary condition at the wall. The method is mathematically exact and has an advantage of having no explicit streamwise gradient terms. It is successfully applied to two different experimental plane turbulent wall jet datasets for which independent estimates of wall shear stress were known. Complications owing to experimental inaccuracies in determining wall shear stress from the proposed method are also discussed.

5:24PM E14.00004 Turbulent Dispersion of Film Coolant in a Turbine Vane Cascade[1] , SAYURI YAPA, CHRISTOPHER ELKINS, JOHN EATON, Stanford University — Gas turbine engines operate at peak temperatures in excess of the material limits because the high pressure turbine nozzles and buckets are film cooled. The nozzle vanes of the first stage turbine use the most cooling air because they are exposed directly to the high temperature combustor exhaust. Existing turbine analysis assumes a uniform temperature at the rotor inlet. However, the coolant does not mix completely with the mainstream flow before impinging on the turbine rotor, and the coolant streaks create variations in temperature along the leading edge of the downstream turbine blades. 3D velocity and concentration measurements are made using magnetic resonance (MR) imaging techniques to study turbulent mixing in a realistic film-cooled nozzle vane cascade. A scalar mixing analogy for thermal diffusion is used in which a chemical contaminant plays the role of temperature. In a typical experiment, the mainstream flow is water and the film coolant is a copper sulfate solution. The concentration of copper sulfate measured anywhere in the flow is a surrogate for normalized temperature. The turbulent scalar diffusivity in the scalar transport equation can be estimated from the MR data and used to improve computational fluid dynamics models.

[1]Army Research Office

5:37PM E14.00005 Turbulent inflow and wake of a marine hydrokinetic turbine, including effects of wave motion , TOBY DEWHURST, MATTHEW ROWELL, JUDSON DECEW, KEN BALDWIN, ROB SWIFT, MARTIN WOSNIK, University of New Hampshire — A research program to investigate the spatio-temporal structure of turbulent flows relevant to marine hydrokinetic (MHK) energy conversion, including turbulent inflow and turbine wakes, has been initiated at UNH. A scale model MHK turbine was deployed from a floating platform at two open water tidal energy test sites, one sheltered (Great Bay Estuary, NH) and one exposed (Muskeget Channel, MA). The inflow upstream of the turbine under test was characterized using an acoustic Doppler Velocimeter (ADV) and an acoustic Doppler current profiler (ADCP), which vary considerably in temporal and spatial resolution as well as practical applicability in this environment. The turbine was operated at previously determined peak efficiency for a given tidal current. The wake of the turbine was measured with a second, traversing ADV during ramp-up and at peak tidal current velocities, at two to six shroud diameters downstream. An inertial motion unit installed near the turbine hub is used to correct for platform motion. A platform-mounted wave-staff and an independently taut-moored pressure sensor were used to measure wave climate. Together, these data are used to validate theoretical and tank model results for utilizing surface-based platforms for MHK turbine deployments.

Sunday, November 18, 2012 4:45PM - 5:37PM –

Session E15 Biofluids: Large Swimmers II 28A - Chair: Hossein Haj-Hajiri, University of Virginia

4:45PM E15.00001 Passive synchronization of finite dipoles in a doubly periodic domain , ALAN CHENG HOU TSANG, EVA KANSO, University of Southern California — We consider the interaction dynamics of finite dipoles in a doubly periodic domain. A finite dipole is a pair of equal and opposite strength point vortices separated by a finite distance throughout its time evolution. The finite dipole dynamical system has been proposed as a model that captures the far-field hydrodynamics interactions in fish schools or collections of swimming bodies in an inviscid fluid. In this work, we formulate the equations of motion governing the dynamics of finite dipoles in a doubly periodic domain. We show that a single dipole in a doubly-periodic box exhibits either regular or chaotic behavior, depending on the initial angle of orientation of the dipole. In the case of the two dipoles, we identify a variety of interesting interaction modes including collision, switching, and passive synchronization of the dipoles. In the case of three dipoles, we observe the formation of relative equilibrium in finite time when the dipoles move together in a way reminiscent to that of flocking behavior.

4:58PM E15.00002 Optimal schooling formations using a potential flow model , ANDREW TCHIEU, MATTIA GAZZOLA, ETH Zurich, ALEXIA DE BRAUER, Université Bordeaux 1, PETROS KOUMOUTSAKOS, ETH Zurich — A self-propelled, two-dimensional, potential flow model for agent-based swimmers is used to examine how fluid coupling affects schooling formation. The potential flow model accounts for fluid-mediated interactions between swimmers. The model is extended to include individual agent actions by means of modifying the circulation of each swimmer. A reinforcement algorithm is applied to allow the swimmers to learn how to school in specified lattice formations. Lastly, schooling lattice configurations are optimized by combining reinforcement learning and evolutionary optimization to minimize total control effort and energy expenditure.

5:11PM E15.00003 Modeling of flapping-fin propulsion with Stuart-Landau oscillator equation[1] , AREN M. HELLUM[2], PROMODE R. BANDYOPADHYAY, Naval Undersea Warfare Center, Newport, RI — Recently, the lowest order thrust measurements in an abstracted twisting and flapping fin have been modeled using a van der Pol-like oscillator (*JFM* **702,** 298-331). A Stuart-Landau oscillator is used here as a higher order model of the interaction of the low aspect ratio flapping fin with its downstream thrust-producing reverse Karman vortex street. "Quasi-steady" equations for the forces produced on flapping fins or wings by the surrounding fluid assume that the lift and drag coefficients are based on '$a_g(t)$', a time-variable angle of attack based on the fin's instantaneous position and velocity. In this work, a wake-modified angle of attack 'a(t)' is used, such that '$a = a_g + a_w$' where '$a_w(t)$' is based on the circulation in the wake. This modification of the geometric angle of attack 'a_g' is justified generally by the conservation of circulation within the fin-wake system, and we argue that a Stuart-Landau oscillator represents a good approximation of the circulation within the wake. Results of this modeling are compared with experimental data taken on the abstracted penguin wing planform; a strong quantitative agreement exists between the experimental and modeled systems. We also model the effects of Reynolds number and the dependence of system oscillation lock-in on initial condition.

[1]Sponsor: ONR34
[2]ONR-ASEE Post-Doctoral Program

5:24PM E15.00004 Role of Strouhal number (St) in free swimming[1] , MEHDI SAADAT, HOSSEIN HAJ-HARIRI, University of Virginia — St of 0.2-0.4 has become synonymous with efficient self propulsion. Is it the cause, or an effect? As has been argued by a number of authors, St alone is insufficient to decide optimal motion because many inefficient combinations of amplitude and frequency lead to the same St. In this talk we show a simple ramification of free swimming where the swim speed and St are outputs. The iso-lines for speed, St, and thrust coincide so long as there is no massive leading-edge separation. It appears that St is simply related to how the drag coefficient and geometry of the body relates to the thrust coefficient and geometry of the propulsor. 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.

[1]Supported by ONR MURI

Sunday, November 18, 2012 4:45PM - 5:50PM –

Session E16 Biofluids: Cardiovascular Simulations and Devices 28B - Chair: Patrick McGah, University of Washington

4:45PM E16.00001 Experimental Flow Characterization of a Flow Diverting Device , EPH SPARROW, RICKY CHOW, U. of Minn., GARY CAMPBELL, Lake Region Medical, AFSHIN DIVANI, U. of Minn., JIAN SHENG, Texas Tech U. — Flow diverters, such as the Pipeline Embolization Device, are a new class of endovascular devices for the treatment of intracranial aneurysms. While clinical studies have demonstrated safety and efficacy, their impact on intra-aneurysmal flow is not confirmed experimentally. As such, optimization of the flow diversion behavior is not currently possible. A quasi-3D PIV technique was developed and applied in various glass models at Re = 275 and 550 to determine the changes to flow characteristics due to the deployment of a flow diverter across the aneurysm neck. Outcomes such as mean velocity, wall shear stress, and others metrics will be presented. Glass models with varying radii of curvature and aneurysm locations will be examined. Experiments were performed in a fully index-matched flow facility using $\sim$10μm diameter polystyrene particles doped with Rhodium 6G dye. The particles were illuminated with a 532nm laser sheet and observed with a CCD camera and a 592nm +/- 43nm bandpass filter. A quasi 3D flow field was reconstructed from multiple orthogonal planes (spaced 0.4mm apart) encompassing the entire glass model. Wall stresses were evaluated from the near-wall flow viscous stresses.

4:58PM E16.00002 Transitional Flow in an Arteriovenous Fistula: Effect of Wall Distensibility[1] , PATRICK MCGAH, DANIEL LEOTTA, KIRK BEACH, ALBERTO ALISEDA, University of Washington — Arteriovenous fistulae are created surgically to provide adequate access for dialysis in patients with end-stage renal disease. Transitional flow and the subsequent pressure and shear stress fluctuations are thought to be causative in the fistula failure. Since 50% of fistulae require surgical intervention before year one, understanding the altered hemodynamic stresses is an important step toward improving clinical outcomes. We perform numerical simulations of a patient-specific model of a functioning fistula reconstructed from 3D ultrasound scans. Rigid wall simulations and fluid-structure interaction simulations using an in-house finite element solver for the wall deformations were performed and compared. In both the rigid and distensible wall cases, transitional flow is computed in fistula as evidenced by aperiodic high frequency velocity and pressure fluctuations. The spectrum of the fluctuations is much more narrow-banded in the distensible case, however, suggesting a partial stabilizing effect by the vessel elasticity. As a result, the distensible wall simulations predict shear stresses that are systematically 10-30% lower than the rigid cases. We propose a possible mechanism for stabilization involving the phase lag in the fluid work needed to deform the vessel wall.

[1]Support from an NIDDK R21 - DK08-1823

5:11PM E16.00003 A CFD study of steady flow of a Newtonian and non-Newtonian fluid through a mildly curved tube with stent-like wall protrusions patterns , CHEKEMA PRINCE, SEAN D. PETERSON, University of Waterloo — Early stent designs caused vessel straightening post-implantation and motivated the exploration of flow in the stent vicinity using straight pipe models with stent-like protrusion patterns. Recent advancements in stent design allow the device to better conform to the native vessel curvature. The present study focuses on the investigation of steady flow through mildly curved pipes with protrusion patterns that emulate current stent designs using computational fluid dynamics (CFD). The modeled geometries include various protrusion frequencies, heights, and widths with flow behavior within the range of physiologically relevant Dean numbers. The results are compared to smooth wall curved pipe models as well as straight pipe protrusion studies. Differences in flow behavior pre/post stent implantation will be discussed. Particular attention will be paid to flow characteristics, such as wall shear stress (WSS) magnitude and WSS gradients, indicative of potential stent failure. Newtonian and non-Newtonian fluid models will be utilized to discuss their impact on flow patterns. The study findings can be used to optimize stent design to mitigate flow conditions associated with stent failure.

5:24PM E16.00004 In vitro characterization of the technique of portal vein embolization by injection of a surgical glue , ANNE-VIRGINIE SALSAC[1], MIHAI-CRISTINEL SANDULACHE, OCEANE LANCON, KHADIJA EL KADRI BENKARA[2], Universite de Technologie de Compiegne — Partial embolization of the portal trunk by glue injection is a minimally invasive technique used in the case of malignant liver tumors. It is conducted few weeks prior to partial liver ablation, when the volume of the remnant liver section is too small to allow hepatectomy. The limitation of glue embolotherapy is that its clinical practice is based on empirical knowledge. The present objective is to study glue injection in a confined blood flow and investigate how the injection dynamics is coupled with glue polymerization. We first characterize polymerization under static conditions for various glue concentrations and then consider the influence of injection. An experimental setup reproduces the co-flow of two immiscible fluids. The glue mixture is injected through a capillary tube into a saline solution with the same ionic concentration, pH and viscosity as blood, flowing steadily in a straight cylindrical tube. The injected phase is visualized with a high-speed imaging system and results are compared with those obtained for non-reacting fluids. These experiments have enabled us to characterize the characteristic times of polymerization of the glue mixtures under static and dynamic conditions and understand how they affect the flow topology of the glue once injected.

[1]Biomechanics and Bioengineering Laboratory (UMR CNRS 7338)
[2]Electromechanical Laboratory (EA 1006)

5:37PM E16.00005 Endovascular Treatment of Thoracic Aortic Dissection: Hemodynamic Shear Stress Study[1] , YIK SAU TANG, SIU KAI LAI, Department of Mechanical Engineering, University of Hong Kong, STEPHEN WING KEUNG CHENG, Department of Surgery, Li Ka Shing Faculty of Medicine, University of Hong Kong, KWOK WING CHOW, Department of Mechanical Engineering, University of Hong Kong — Thoracic Aortic Dissection (TAD), a life threatening cardiovascular disease, occurs when blood intrudes into the layers of the aortic wall, creating a new artificial channel (the false lumen) beside the original true lumen. The weakened false lumen wall may expand, enhancing the risk of rupture and resulting in high mortality. Endovascular treatment involves the deployment of a stent graft into the aorta, thus blocking blood from entering the false lumen. Due to the irregular geometry of the aorta, the stent graft, however, may fail to conform to the vessel curvature, and would create a "bird-beak" configuration, a wedge-shaped domain between the graft and the vessel wall. Computational fluid dynamics analysis is employed to study the hemodynamics of this pathological condition. With the 'beaking' configuration, the local hemodynamic shear stress will drop below the threshold of safety reported earlier in the literature. The oscillating behavior of the shear stress might lead to local inflammation, atherosclerosis and other undesirable consequences.

[1]Supported by the Innovation and Technology Fund of the Hong Kong Government.

Sunday, November 18, 2012 4:45PM - 6:03PM –

Session E17 Biofluids: Plants 28C - Chair: Dwight Whitaker, Pomona College

4:45PM E17.00001 Scalings of drag reduction by elastic or brittle reconfiguration in plants , EMMANUEL DE LANGRE, SEBASTIEN MICHELIN, DIEGO LOPEZ, LadHyX - Ecole polytechnique — Slender flexible structures such as plants are elastically deformed by external flows. When the deformation is large , this results in a significant reduction of drag. We give a theoretical value of the exponent of the dependence on flow velocity in the drag law, based on scaling arguments. The theoretical value is shown to compare well with experimental data on a very large variety of plants, ranging from full trees to aquatic vegetation. It is also shown that elastic reconfiguration affects more the evolution of local bending stress or the uprooting moment than the total drag. Moreover, a nonlinearity in the local elastic behavior does not affect significantly the exponent of drag. The approach is then generalized to the case of brittle reconfiguration, or flow-induced pruning, a mechanism by which plant avoid permanent base damage under flow by losing parts of their architecture.

4:58PM E17.00002 Morphological changes in kelp result in reduced drag and increased stability , JEFFREY ROMINGER, HEIDI NEPF, Massachusetts Institute of Technology — Many species of kelp change their blade morphology in response to the local flow environment. Several studies document increases in blade thickness, and thus increases in blade rigidity, in more energetic flow environments. This morphological strengthening has been traditionally understood to provide increased strength in tension which allows blades to remain intact under high tensile forces. In this work, however, we describe two mechanisms by which increases in blade thickness can also reduce fluid drag forces: first, by reducing the drag created by blade perturbations provoked by passing turbulence; second, by increasing the blade stability in the fluid-elastic stability space. To describe the stable interactions between the blade and turbulent flow, we appeal to previous experimental results that draw on Lighthill's elongated body theory to explain the drag force benefits of increased rigidity. As blades pass the fluid-elastic stability threshold and become unstable, there is a well-documented increase in the drag coefficient of over one order of magnitude. Therefore, morphological changes in blade thickness not only increase strength in tension, but can help reduce dynamic drag forces associated with blade bending.

5:11PM E17.00003 Inertial particle transport from peat moss vortex rings , SAMUEL WHITEHEAD, EMILY CHANG, DWIGHT WHITAKER, Pomona College — We present a numerical analysis of how vortex rings from *Sphagnum* moss disperse their spores. Comparisons of the results of our CFD model with data measured from high-speed video reveal that the pressure inside the capsules is only 2 atm, which is significantly less than has been reported in the literature. Moreover vortex rings produced by these pressures do not optimize the impulse to the fluid as is seen in other biological systems. Here we present an analysis of the efficiency with which vortex rings from *Sphagnum* transport spores to heights where they can be carried by wind currents. Spore trajectories determined from a modified Maxey-Riley equation form a dynamical system for which we find the Lagrangian coherent structures (LCS), which define regions where spores are entrained in the vortex bubble. By analyzing the dependence of the LCS on spore size, capsule pressure, and morphology we will assess the efficiency with which vortex rings from *Sphagnum* transport spores.

5:24PM E17.00004 Aerodynamics of puffball mushroom spore dispersal , GUILLERMO AMADOR, ALEX BARBERIE, DAVID HU, Georgia Institute of Technology — Puffball mushrooms $Lycoperdon$ are spherical fungi that release a cloud of spores in response to raindrop impacts. In this combined experimental and theoretical study, we elucidate the aerodynamics of this unique impact-based spore-dispersal. We characterize live puffball ejections by high speed video, the geometry and elasticity of their shells by cantilever experiments, and the packing fraction and size of their spores by scanning electron microscope. We build a dynamically similar puffball mimic composed of a tied-off latex balloon filled with baby powder and topped with a 1-cm slit. A jet of powder is elicited by steady lateral compression of the mimic between two plates. The jet height is a bell-shaped function of force applied, with a peak of 18 cm at loads of 45 N. We rationalize the increase in jet height with force using Darcy's Law: the applied force generates an overpressure maintained by the air-tight elastic membrane. Pressure is relieved as the air travels through the spore interstitial spaces, entrains spores, and exits through the puffball orifice. This mechanism demonstrates how powder-filled elastic shells can generate high-speed jets using energy harvested from rain.

5:37PM E17.00005 Effect of multi-ions on active flow regulation in plants , JEONGEUN RYU, Department of Mechanical Engineering, POSTECH, SUNGSOOK AHN, Center for Biofluid and Biomimic Research, POSTECH, SEUNG-GON KIM, Department of Mechanical Engineering, POSTECH, HWASUK OH, TAEJOO KIM, KAERI (The Korea Atomic Energy Research Institute), SANG JOON LEE, Department of Mechanical Engineering & Division of Integrative Biosciences and Biotechnology, POSTECH — Plants have been known to regulate ion-mediated flows actively in xylem vessels. Pits, the porous structures in xylem vessels, are presumed to play a key role in the ion-mediated flow regulation based on dynamic swelling and shrinking of their pectic matrix. However, the autonomous flow regulation in plants has not been elucidated yet and the pectin-swelling hypothesis seems to be simply applied to account for dynamic modulation of xylem conductance. In this study, the effects of multiple ions and their concentration on the water transport in plants were experimentally investigated. In addition, the active regulation mechanism of xylem water flow was also examined with considering the ionic effect.

5:50PM E17.00006 How the Venus flytrap actively snaps: hydrodynamic measurements at the cellular level , MATHIEU COLOMBANI, YOEL FORTERRE, IUSTI CNRS UMR 7343, Aix-Marseille Univ., France, GEP TEAM — Although they lack muscle, plants have evolved a remarkable range of mechanisms to create rapid motion, from the rapid folding of sensitive plants to seed dispersal. Of these spectacular examples that have long fascinated scientists, the carnivorous plant Venus flytrap, whose leaves snap together in a fraction of second to capture insects, has long been a paradigm for study. Recently, we have shown that this motion involves a snap-buckling instability due to the shell-like geometry of the leaves of the trap. However, the origin of the movement that allows the plant to cross the instability threshold and actively bend remains largely unknown. In this study, we investigate this active motion using a micro-fluidic pressure probe that gives direct hydraulic and mechanical measurements at the cellular level (osmotic pressure, cell membrane permeability, cell wall elasticity). Our results challenge the role of osmotically-driven water flows usually put forward to explain Venus flytrap's active closure.

Sunday, November 18, 2012 4:45PM - 5:50PM –

Session E18 Biofluids: Vesicles 28D - Chair: Michael Miksis, Northwestern University

4:45PM E18.00001 Lipid Bilayer Vesicle Dynamics in DC Electric Fields , LANE MCCONNELL, Northwestern University, PETIA VLAHOVSKA, Brown University, MICHAEL MIKSIS, Northwestern University — Vesicles, which are closed lipid bilayers, provide a valuable model to study the dynamics of the biomembranes that surround cells. Recent small deformation analysis of vesicles exposed to a DC electric field has revealed several interesting phenomena, including transitions from oblate to prolate ellipsoidal shapes and damped tumbling in the case of combined electric field and shear flow. Here we investigate the behavior and stability of a vesicle in a uniform DC electric field numerically using the boundary integral method. The vesicle membrane is modeled as an infinitely thin, capacitive, area-incompressible interface, with the surrounding fluids presumed to act as leaky dielectrics. Vesicle dynamics are determined by balancing the hydrodynamic, bending, tension, and electric stresses on the membrane. Our investigation compares the full nonlinear numerical results to the small deformation theory and to recent experimental data, and presents a thorough analysis of the relevant parameter space.

4:58PM E18.00002 Asymmetric Vesicle Instability in Extensional Flow[1] , ANDREW SPANN, HONG ZHAO, ERIC SHAQFEH, Stanford University — Previous researchers have chronicled the breakup of drops in an extensional flow as they stretch into a dumbbell shape with a long thin neck. Motivated by recent experimental observations, we study an apparently similar problem with vesicles, which are deformable but incompressible membranes that conserve area and volume. First, we simulate vesicles in an unbounded uniaxial extensional flow which are given general radial perturbations from an initially stable symmetric equilibrium state. For sufficiently low reduced volume (< 0.74 at matched inner/outer viscosity) there exists a capillary number at which an asymmetric perturbation mode will grow, resulting in the formation of an asymmetric dumbbell shape with a thin connecting cylindrical bridge analogous to the shapes associated with drop breakup. Our simulations help elucidate a mechanism for this instability based on a competition between internal pressure differentials in the vesicle resulting from the membrane bending force and ambient flow. We compare and contrast this transition to the "standard" drop breakup transition.

[1]Funded by NSF GRFP and Stanford Graduate Fellowship

5:11PM E18.00003 A transient solution and scaling laws for vesicle electrodeformation and relaxation[1] , HAO LIN, JIA ZHANG, JEFFEREY ZAHN, Rutgers University — A transient analysis for vesicle deformation and relaxation under DC electric fields is presented. The theory extends from a droplet model developed by us, with the additional consideration of a lipid membrane separating two fluids of arbitrary properties. For the latter, both a membrane-charging and a membrane-mechanical model are supplied. The main result is an ODE governing the evolution of the vesicle aspect ratio. The model prediction is extensively compared with experimental data, and is shown to accurately capture the system behavior. More importantly, the comparison reveals that vesicle relaxation obeys a universal behavior regardless of the means of deformation. The process is governed by a single timescale that is a function of the vesicle initial radius, the fluid viscosity, and the initial membrane tension. This universal scaling law can be used to calculate membrane properties from experimental data.

[1]NSF award CBET-0747886

5:24PM E18.00004 The dynamics of a vesicle during adhesion processes[1] , MAURICE BLOUNT, MICHAEL MIKSIS, STEPHEN DAVIS, Northwestern University — We analyze the adhesion of a two-dimensional vesicle to a flat substrate by a long-range attractive, short-range repulsive force, in the asymptotic limit that the length scale on which this force acts is much smaller than the vesicle's perimeter. As the vesicle is pulled down towards the substrate, a thin wetting layer is trapped underneath it whose thickness is determined by the adhesive force. At the edges of this wetting layer are boundary layers whose evolution is governed by adhesive, bending and viscous stresses. We use a lubrication approximation to describe the fluid flow inside these boundary-layer regions, and we show how these regions control the dynamics of the remainder of the vesicle. We obtain traveling-wave solutions for the lubrication flow and discuss their relevance during the adhesive process.

[1]Supported by NIH Grant 5R01GM086886

5:37PM E18.00005 The dynamics of adhesion of a pair of vesicles , JOHANN WALTER, L. GARY LEAL, University of California, Santa Barbara, Dept. of Chemical Engineering — Adhesive interactions within a suspension of vesicles, such as many personal care products, vectors for drug delivery or artificial blood, can lead to aggregation of the vesicles and dramatic changes to the properties of the suspension. We study the adhesion of a pair of unilamellar, charged vesicles under flow, in the presence of a non-adsorbing polymer or micelle creating a depletion attraction force between the vesicles. 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 (resistance to bending and area increase are both taken into account). The dynamics of the drainage process are studied. At steady state, the adhesion energy is found to depend greatly on the ability of the vesicles to increase their surface area. Finally, when the vesicles are separated in an elongational flow, different behaviors are observed depending on the deformability of the vesicles: an increase of the film thickness with a constant contact area, or peeling-off phenomenon where the contact area decreases at constant film thickness.

Sunday, November 18, 2012 4:45PM - 5:50PM –

Session E19 Thin Films 28E - Chair: Randy Ewoldt, University of Illinois at Urbana-Champaign

4:45PM E19.00001 Interaction model between a liquid film and a spherical probe , RENE LEDESMA ALONSO, DOMINIQUE LEGENDRE, PHILIPPE TORDJEMAN, Institut de Mécanique des Fluides de Toulouse (IMFT) — To find a liquid surface profile, when performing AFM measurements, probe interaction effects should be identified. Herein, the behavior of a liquid film free surface (thickness E, surface tension γ and density difference $\Delta\rho$), disposed over a flat surface and in the presence of a spherical probe (radius R) is forecast. A bump-like surface shape is observed, due to the probe/film interaction (characterized by the Hamaker constant H_{pl}). In addition, the attraction between the film and the substrate (depicted by H_{sl}) opposes the axial and radial deformation ranges. Several parameters portray the equilibrium shape: Bond $B_o = (\Delta\rho g R^2)/\gamma$ and modified Hamaker $H_a = 4H_{pl}/(3\pi\gamma R^2)$ numbers, Hamaker ratio $A = H_{ls}/H_{pl}$, separation distance D/R and film thickness E/R. We focus on the effect of geometry, nevertheless special attention is given to the role of physical parameters. Employing an augmented Young-Laplace equation, the equilibrium profile is described by a strongly non-linear ODE. A critical distance, below which the irreversible wetting process of the spherical probe occurs, is predicted. Our results provide simple relationships between parameters, which determine the optimal scanning conditions over liquid films.

4:58PM E19.00002 Dynamics of precursor films , MARK FRANKEN, CHRISTIAN POELMA, JERRY WESTERWEEL, Laboratory for Aero & Hydrodynamics, Delft University of Technology — The precursor film, formed ahead of a macroscopic droplet edge during spreading, is studied experimentally using a technique based on Total Internal Reflection Fluorescence Microscopy. This microscopy technique uses an evanescent wave resulting from total internal reflection of incident light to illuminate a fluorescent dye. The technique has sufficient spatial as well as temporal resolution, allowing us to measure the precursor film for different contact line velocities and liquids. We find that the precursor film thickness h(x) scales as 1/sqrt(x) and is independent of contact line velocity (hence capillary number). These results confirm for the very first time the theoretical predictions based on slip at the solid surface.

5:11PM E19.00003 Lubrication analysis of the nanometric coating film deposited during gravure printing , UMUT CEYHAN, RUNGROT KITSOMBOONLOHA, S.J.S. MORRIS, VIVEK SUBRAMANIAN, University of California, Berkeley — We report the importance of doctor blade-tip's geometry and wettability on the formation of coating film of thickness 1-10 nm after wiping of the excess ink used for gravure printing of electronics. Several authors have worked on the blade coating problem, addressing elastohydrodynamic effects; however, the coating film deposited during gravure printing is about 3 orders of magnitude thinner than micrometer scale created in blade coating. The blade-tip radius is consequently large compared with the film and gap thickness, allowing the blade tip to be approximated by a parabola. Hydrodynamic forces are concentrated within this inner region. In the gap entry, streamlines converge making the pressure large and positive; downstream, streamlines diverge making pressure large, but negative. This large negative pressure affects the coating film thickness by tending to draw the meniscus back into the narrow gap. Gap thickness and coating film thickness are determined as part of the solution of a free-boundary problem: we couple lubrication analysis of the gap flow in the gap to Landau-Levich analysis of the film flow. The resultant hydrodynamic force and couple exerted within the inner region are compared with those exerted on the outer portion of the blade and parameters affecting the solution of the problem on the coating film formation are examined in detail.

5:24PM E19.00004 Experimental study of the residue film in direct gravure printing of electronics , RUNGROT KITSOMBOONLOHA, UMUT CEYHAN, S.J.S. MORRIS, VIVEK SUBRAMANIAN, University of California, Berkeley — Direct gravure printing is a promising candidate for high resolution printing of electronics. During gravure printing, excess ink is removed from a patterned plate by a doctor blade and a residue film ∼10nm thick is left on the non-patterned area. This residue film degrades the pattern fidelity and has to be minimized. To understand the issues involved, we performed experiments on this residue film using a custom printer. We investigated the effects of wettability, ink viscosity and printing speed to understand the mechanisms of residue film formation. The results reveal that there are two types of residue film, originating from the wettability of the printing components and elastohydrodynamics. The first is a cell-dependent residue film, produced by the doctor blade dragging ink out from engraved cells. The second is a uniform residue film, which is caused by an increase in the gap between the patterned plate and doctor blade due to elastohydrodynamic pressure. As the capillary number increases, the cell-dependent residue film decreases, while the uniform residue film increases. These two types of residue films force gravure printing to be operated within a regime covering carefully controlled wettability, ink viscosity and printing speed, where the residue film is minimized.

5:37PM E19.00005 Nanometer-scale free surface flow of molten polyethylene from a heated atomic force microscope tip , RANDY EWOLDT, JONATHAN FELTS, SUHAS SOMNATH, WILLIAM KING, Department of Mechanical Science and Engineering, University of Illinois Urbana-Champaign, Urbana, IL 61801, USA — We experimentally investigate nanometer-scale free surface flow of molten polyethylene from a heated atomic force microscope (AFM) cantilever, a nanofabrication process known as thermal dip-pen nanolithography (tDPN). Fluid is deposited from the AFM tip onto non-porous substrates whether the tip is moving or fixed. We find that polymer flow depends on surface capillary forces and not on shear between tip and substrate. The polymer mass flow rate is sensitive to the temperature-dependent polymer viscosity. Additionally, the flow rate increases when a temperature gradient exists between the tip and substrate. We hypothesize that the polymer flow is governed by thermal Marangoni forces and non-equilibrium wetting dynamics caused by a solidification front within the feature.

Sunday, November 18, 2012 4:45PM - 5:50PM –

Session E20 Turbulent Boundary Layers III: Theory 30A - Chair: Mihailo Jovanovic, University of Minnesotta

4:45PM E20.00001 Dynamics of Streak and Roll Structures in Turbulent Plane Couette Flow , DENNICE GAYME, Johns Hopkins University, BRIAN FARRELL, Harvard University, PETROS IOANNOU, University of Athens, BINH LIEU, MIHAILO JOVANOVIC, University of Minnesota — The dominance of streamwise elongated features in transition and turbulence motivates the use of streamwise constant (so called 2D/3C) and statistical mean state dynamics models to investigate the dynamics of the well studied streamwise elongated roll and streak structures in plane Couette flow. Predictions resulting from two such theoretical models are compared to those of fully resolved DNS data. The 2D/3C model correctly captures the roll/streak structures, the momentum transfer mechanism and statistical features of turbulence such as the mean turbulent velocity profile. Furthermore, the statistical mean state dynamics model, which couples the streamwise constant mean flow and (streamwise varying) perturbation dynamics, adds the critical feedback mechanism arising from the important $k_x \neq 0$ information to this streamwise constant framework. The resulting model not only predicts the roll/streak structures, mean profile and momentum transfer but also exhibits a bifurcation from the stable 2D/3C dynamics to a self-sustaining turbulent state that closely resembles that seen in DNS. **Acknowledgements:** This research was performed at the 2012 Stanford CTR Summer Program.

4:58PM E20.00002 Coherent structure, amplitude modulation and higher order statistics in wall turbulence[1] , BEVERLEY MCKEON, California Institute of Technology, ATI SHARMA, University of Sheffield — Coherent structure in wall turbulence is shown to be captured by the frequency-domain treatment of the Navier-Stokes equations as a directional amplifier proposed by McKeon & Sharma (2010). Simple combinations of the predicted response modes (which take the form of radially-varying travelling waves), consistent with the nonlinear triadic interaction known for wavenumber interactions, are offered which minimally predict all types of structures including hairpin vortices and modulated hairpin packets. One such combination is understood to form a turbulence "kernel," which, it is proposed, constitutes a self-exciting process analogous to the near-wall cycle. The phase relationships explain important skewness and correlation results known in the literature. It is shown that the local shear associated with very large scale motions acts to organize hairpin-like structures such that they co-locate with areas of low streamwise momentum. Compelling evidence for the theory is presented based on comparison to observations of structure and statistics reported in the experimental and numerical simulation literature and similarities with other analytical and empirical models are discussed

[1]The support of AFOSR under grant FA 9550-09-1-0701 is gratefully acknowledged.

5:11PM E20.00003 Scaling of normal stresses in the turbulent boundary layer , PETER MONKEWITZ, Swiss federal Institute of Technology Lausanne (EPFL), HASSAN NAGIB, IIT, Chicago, USA — Concentrating on the canonical zero pressure gradient (ZPG) turbulent boundary layer (TBL), different scalings of normal stresses, in particular of $< u^2 >$, have been proposed. In the range of Reynolds numbers where measurements are available, the best data collapse is obtained by scaling stream-wise fluctuations with the free stream velocity U_∞. It is shown with the underlying RANS equations that this choice, together with the traditional Rotta outer scale $\delta^* U_\infty^+$ and the "log law" leads to a boundary layer thickness which decreases in the downstream direction. In other words, if one insists on both the traditional mean flow similarity and on scaling normal stresses with U_∞, all (growing) TBLs seen so far are very far from their true asymptotic (shrinking) state. Alternative assumptions/scalings and their consequences will be discussed.

5:24PM E20.00004 A mean profile formulation for canonical wall-bounded turbulent flows consistent with the mean momentum equation , JIMMY PHILIP, IVAN MARUSIC, Department of Mechanical Engineering, University of Melbourne, JOSEPH KLEWICKI, Department of Mechanical Engineering, University of New Hampshire & Department of Mechanical Engineering, University of Melbourne — The mean velocity profile for wall bounded flows is formulated in a manner that is consistent with the magnitude ordering of terms and characteristic length scales associated with the mean momentum equation. Close to the wall, the viscous length characterizes the dynamics, and Prandtl's law-of-the-wall holds. In an outer inertial region where the dominant balance is between the Reynolds stress gradient and the pressure gradient (or mean advection), the mean flow is most closely approximated by a logarithmic function. The width of this region is (asymptotically) characterized by the outer length scale. As initially demonstrated by Wei et al (2005), for all canonical wall-flows the mean viscous force retains dominant order out to a wall-normal location that, in inner units, is $O(\sqrt{\delta^+})$, where δ^+ is the Karman number. The present formulation respects these known properties. This formulation predicts that for low δ^+ the log-law is approached from "above" the logarithmic line, while for high δ^+ the log-law is attained from "below." These subtle properties and the general functional form are shown to be in very good agreement with the mean velocity data available from boundary layer, pipe and channel flows.

5:37PM E20.00005 A new theory for the streamwise turbulent fluctuations in pipe flow , MARCUS HULTMARK, Princeton University — A new theory for the streamwise turbulent fluctuations, in fully developed pipe flow, is proposed. The new theory, which is based on the near asymptotic analysis introduced by George & Castillo 1997, introduces a sensitivity function for the fluctuations, which is related to the increase of the non-dimensionalized fluctuations with Reynolds number. The theory predicts that the fluctuations will experience a logarithmic behavior if the sensitivity function is constant in a region in space and that the magnitude of the constant will correspond to the slope of the logarithm. The theory extends the similarities between the mean flow and the streamwise turbulence fluctuations, as observed in experimental high Reynolds number data, to also include the theoretical derivation. Experimental data show that a mesolayer, similar to that introduced by Wosnik et al. 2000, exists for the fluctuations for $300 > y^+ > 800$, which coincides with the mesolayer for the mean velocities. In the mesolayer, the flow is still affected by viscosity, which acts to decrease the fluctuations and to form an outer peak in the fluctuation profile.

Sunday, November 18, 2012 4:45PM - 5:50PM –

Session E21 Turbulence Simulation: Compressible/Hypersonic 30B - Chair: Sharath Girimaji, Texas A&M University

4:45PM E21.00001 Direct numerical simulation of compressible Kolmogorov flow , REBECCA BERTSCH, SHARATH GIRIMAJI, GAURAV KUMAR, Texas A&M University — Direct numerical simulations to investigate three-dimensional compressible turbulent Kolmogorov flow using the gas kinetic method are performed. The evolutions of single modes in isolation are examined in order to gain insight into the evolution of statistics of modes in collection. Single modes, or straight modes, are either stream-wise or span-wise and their combinations refer to oblique modes. Oblique modes (initially two-dimensional) are analyzed to isolate unstable modes and study the effect of gradient Mach number and Taylor-microscale Reynolds number. The competition between the Kelvin-Helmholtz and uniform shear instability is observed to determine the effect on flow-thermodynamic interactions. Simulations of turbulent Kolmogorov flow with varying amount of compressibility, determined by the gradient Mach number, are analyzed to support prior results of the stabilizing effect of compressibility in turbulent shear flows.

4:58PM E21.00002 Reynolds and Mach number scaling in compressible isotropic turbulence[1] , SHRIRAM JAGANNATHAN, DIEGO DONZIS, Texas A&M University — A large database of well-resolved Direct Numerical Simulations of forced compressible isotropic turbulence, including recent simulations at 2048^3, with Taylor Reynolds numbers up to 400, and a range of Mach numbers (M_t) is used to investigate Reynolds and Mach number effects in compressible turbulence. The spectrum of dilatational energy is greatly affected by compressibility, unlike the solenoidal part. The density spectrum shows an incipient $k^{-5/3}$ at high Reynolds whose height increases with M_t. The scaling of dissipation and pressure-dilatation, the two exchange mechanisms between kinetic and internal energy, are also investigated. While the dilatational dissipation scales as M_t^2 at low Mach numbers, a stronger effect is observed at high M_t. Although the mean pressure-dilatation, which scales with dilatational dissipation, is small compared to the total dissipation for low Mach numbers, at high Mach numbers it is comparable to the total dissipation. The probability density function of pressure-dilatation shows fluctuations greater than O(100) times the mean indicating localized but strong positive and negative transfers which tend to give only a small net contribution when averaged over space. This effect increases at high Reynolds and low Mach numbers.

[1]The authors gratefully acknowledge the support of NSF.

5:11PM E21.00003 Turbulence statistics with quantified uncertainty in cold-wall supersonic channel flow[1] , RHYS ULERICH, ROBERT D. MOSER, The University of Texas at Austin — To investigate compressibility effects in wall-bounded turbulence, a series of direct numerical simulations of compressible channel flow with isothermal (cold) walls have been conducted. All combinations of Re $= \{3000, 5000\}$ and Ma $= \{0.1, 0.5, 1.5, 3.0\}$ have been simulated where the Reynolds and Mach numbers are based on bulk velocity and sound speed at the wall temperature. Turbulence statistics with precisely quantified uncertainties computed from these simulations will be presented and are being made available in a public data base at http://turbulence.ices.utexas.edu/. The simulations were performed using a new pseudo-spectral code called Suzerain, which was designed to efficiently produce high quality data on compressible, wall-bounded turbulent flows using a semi-implicit Fourier/B-spline numerical formulation.

[1]This work is supported by the Department of Energy [National Nuclear Security Administration] under Award Number [DE-FC52-08NA28615].

5:24PM E21.00004 Strong shock and turbulence interactions w/ or w/o thermochemical non-equilibrium effects[1] , XIAOWEN WANG, PRADEEP S. RAWAT, XIAOLIN ZHONG, University of California, Los Angeles — The underlying physics in shock and turbulence interaction is essential for a better understanding of many natural processes as well as scientific and engineering applications. One of the fundamental building blocks in these complex processes and applications is the canonical problem of isotropic turbulence and normal shock. Unfortunately, even this fundamental problem is not well understood for strong shocks. We have conducted extensive DNS studies on strong shock and turbulence interaction for perfect gas flow with mean Mach numbers ranging from 2 to 30. The results show some new trends in turbulent statistics as mean Mach number is increased. However, gas temperature increases dramatically after strong shocks so that numerical simulations based on perfect gas flow may not be enough. The effects of thermochemical non-equilibrium flow including internal energy excitations, translation-vibration energy relaxation, and chemical reactions among different species need to be considered. We have developed a new high-order shock-fitting solver based on the 5-species air chemistry and recently thermal non-equilibrium models. The code package has been tested and applied to DNS of strong shock and turbulence interactions with thermochemical non-equilibrium effects.

[1]The research was supported by DOE office of Science and XSEDE computer resources.

5:37PM E21.00005 Large Eddy Simulation of Shock / Turbulence Interactions , NATHAN GRUBE, JUSTINE LI, PINO MARTIN, University of Maryland — We assess large eddy simulations (LES) of shock / turbulence interactions in two configurations: shock / isotropic turbulence interaction (SITI) and shock / turbulent boundary layer interaction (STBLI). In the simpler SITI configuration, we assess the performance of the dynamic mixed model (DMM) through comparisons with converged DNS data for incoming Mach numbers ranging from 3.3 to 4.6 and turbulence Mach numbers ranging from 0.5 to 0.8. The highly compressible nature of the incoming turbulence requires the use of robust shock-capturing throughout the entire domain. We use a linearly and nonlinearly optimized WENO method. The same LES code is also assessed on the more complicated STBLI case of a 24° compression ramp in Mach 2.9 flow by comparison with the DNS of Priebe and Martín. (Priebe, Stephan and Martín, M. Pino, "Low-frequency unsteadiness in shock wave-turbulent boundary layer interaction," J. Fluid Mech., vol. 699, pp. 1-49, 2012.)

Sunday, November 18, 2012 4:45PM - 5:50PM –

Session E22 Turbulence Modeling: LES 30C - Chair: James McDonough, University of Kentucky

4:45PM E22.00001 Compressible Explicitly Filtered Large-Eddy Simulation Subgrid-Scale Models Based on the Poor Man's Navier–Stokes Equations , J.M. MCDONOUGH, J.P. STRODTBECK, University of Kentucky — A single-parameter large-eddy simulation (LES) method for compressible flows that combines explicit filtering with a chaotic backscatter term based on the compressible "poor man's" Navier–Stokes (PMNS) equation (Strodtbeck *et al., Int. J. Bifur. and Chaos*, 2012) is presented and compared to direct numerical simulation (DNS) of homogeneous, isotropic, decaying turbulence in a periodic cube with a turbulent Mach number of 0.3 and Taylor micro-scale Reynolds number of 72. The DNS employed a 129^3 mesh, and LES was run on a 65^3 mesh. The backscatter is created by utilizing output from the PMNS equations as a multiplier on the high-pass filtered momentum to induce mixing, then using the results in a single-parameter linear forcing term that is simply added to the solution procedure. It is demonstrated that with a judicious choice of model parameter, the backscatter model is capable of enhancing small-scale turbulent flow structures and improving the flow statistics and spectral characteristics of the LES solution to more closely match the DNS results.

4:58PM E22.00002 Constrained Large Eddy Simulation of Wall-bounded Turbulent Flows with Massive Separations , ZHENHUA XIA, YIPENG SHI, ZUOLI XIAO, SHIYI CHEN, State Key Laboratory for Turbulence and Complex System, College of Engineering, Peking University — Constrained Large-eddy Simulation (CLES) has been recently developed to simulate turbulent flows with massive separations. Different from traditional large eddy simulation (LES) and hybrid RANS/LES approaches, the CLES simulates the whole flow domain by large eddy simulation while enforcing a Reynolds stress constraint on the subgrid-scale (SGS) stress models in the near-wall region. In this paper, we use the CLES method to simulate two separated flows, i.e. flow over periodic-hills and flow around NACA0021 airfoil at 60 degrees angle of attack. The results are compared with those from DES and other traditional simulation methods using the same grid resolution. In flow over periodic-hills, the results show that CLES can capture the mean separation location and reattachment location more accurately. In flow around NACA0021 airfoil at 60° angle of attack, CLES can estimate the pressure coefficients after separations more precisely. Furthermore, the computational cost of the CLES is almost the same as that of DES. We will also discuss the application of CLES for aerodynamic.

5:11PM E22.00003 Nonequilibrium energy spectrum in subgrid-scale one-equation model in LES , KIYOSI HORIUTI, TAKAHIRO TAMAKI, Tokyo Institute of Technology — The subgrid-scale (SGS) modeling in LES which accounts for effect of unsteadiness and nonequilibrium state is considered by employing the transport equation for the SGS energy (one-equation model). Perturbation expansion about the Kolmogorov -5/3 energy spectrum which constitutes a base equilibrium state in the inertial subrange yields -7/3 spectrum. These spectra are extracted in the DNS data, and their roles in generation of energy cascade have been revealed. The SGS energy spectrum which governs one-equation model is sought in a perturbative manner. Besides the base -5/3 spectrum assumed in the Smagorinsky model, -7/3 power component is derived, which is induced by temporal variations of SGS energy. We propose the nonequilibrium Smagorinsky model in which estimate of the SGS energy based on the -7/3 spectrum is added to the Smagorinsky model. Assessment in forced homogeneous isotropic turbulence showed that performance of the Smagorinsky and one-equation models for prediction of temporal variations of turbulence energy is not satisfactory, whereas improvement is achieved in the new model. This occurred because natural continuation of grid-scale spectrum, which contains both -5/3 and -7/3 components, into the SGS and associated energy transfer is established in the new model.

5:24PM E22.00004 ABSTRACT WITHDRAWN –

5:37PM E22.00005 ABSTRACT WITHDRAWN –

Sunday, November 18, 2012 4:45PM - 5:50PM –

Session E23 Turbulence Theory: 2D Turbulence 30D - Chair: Marie Farge, LMD-IPSL-CNRS ENS

4:45PM E23.00001 Spatial structure of spectral transport in two-dimensional flow[1] , YANG LIAO, NICHOLAS OUELLETTE, Yale University — Recently developed tools based on filtering have begun to allow the spatial localization of spectral activity in turbulent flow. These filter-space techniques (FSTs) have been used, for example, to study the mechanisms responsible for the double cascade in two-dimensional turbulence or the coherence of the spectral energy flux along Lagrangian trajectories. But FSTs can sometimes give results that seem potentially spurious, such as very large signals around vortex cores. Using both a simple analytical model flow field and measurements from a quasi-two-dimensional experimental flow, we study the results of FSTs in detail. In particular, we show that a classic decomposition of the spectral energy flux may be fruitful.

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

4:58PM E23.00002 Fokker-Planck description of the inverse cascade in two-dimensional turbulence , OLIVER KAMPS, Center for Nonlinear Science, University of Muenster, Muenster, Germany, MICHEL VOSSKUHLE, Laboratoire de Physique, Ecole Normale Superieure de Lyon, Lyon, France — In many approaches the mathematical description of fully developed turbulence relies on the statistical properties of the longitudinal velocity increments $\xi(r) = U(x+r) - U(x)$. In [1] the increment statistics is described as a Markov process in scale, leading to a Fokker-Planck description of the probability density functions (PDFs) for the velocity increments. The universality of this approach was tested for different kinds of three-dimensional flows like inhomogeneous turbulence, fractal grid generated turbulence and for the transition of a flow from a vortex street to fully developed turbulence in a cylinder wake the flow. In this talk we want to extend the test for the universality of the Markov description by analyzing data from numerical simulations of the inverse energy cascade in two-dimensional turbulence. The central question is whether the velocity field of the inverse cascade can be modeled as Markov process in scale similar to the three-dimensional case. By estimating the coefficients of the Fokker-Planck equation we are able to discuss the role of intermittency and differences to three-dimensional flows.

[1] Friedrich R., Peinke J., *Phys. Rev. Lett, vol. 78, pp. 863-866 (1997)*

5:11PM E23.00003 Simple invariant solutions embedded in 2D Kolmogorov turbulence[1] , RICH KERSWELL, GARY CHANDLER, Bristol University — Ideas from dynamical systems have recently provided fresh insight into transitional and weak turbulent flows where the system size is smaller than the spatial correlation length. Viewing such flows as a trajectory through a phase space littered with invariant solutions and their stable and unstable manifolds has proved a fruitful way of understanding such flows. It is therefore natural to ask whether any ideas attempting to rationalise chaos may have something to say about developed turbulence. One promising line of thinking in low-dimensional, hyperbolic dynamical systems stands out as a possibility - Periodic Orbit Theory (Auerbach et al 1987, Cvitanovic 1988 and the review by Lan 2010). I will discuss an attempt to apply this in 2D Kolmogorov turbulence: body-forced flow (where the forcing is monochromatic and large scale) over a doubly periodic box.

[1]Research supported by EPSRC.

5:24PM E23.00004 Exponential decay of a passive tracer variance in a two-dimensional Navier-Stokes flow , FARID AIT CHAALAL, California Institute of Technology — We study numerically the decay of a passive tracer in a dynamically consistent flow solution of the two-dimensional Navier-Stokes equation and in the limit of small diffusion. We observe that the decay of the variance becomes quickly exponential, as previously observed in simple chaotic maps, like the well-studied renewing sine flow proposed by Pierrehumbert in the early nineties. However, after a few tens of large-eddy turnover times, the decay rate changes. We interpret this result in light of theories developed for mixing in simple ergodic flows, in particular local Lagrangian stretching theories. It is found that they only can explain very partially the phenomenology we observe. In particular, they cannot capture the rôle of coherent vortices which is crucial, particularly in the very long-term decay, and might explain the decay rate change.

5:37PM E23.00005 Comparison of turbulent mixing and chaotic advection in a two-dimensional wall-bounded domain , BENJAMIN KADOCH, IUSTI-CNRS, Aix-Marseille University, Marseille, France, WOUTER BOS, LMFA-CNRS, Ecole Centrale de Lyon, University of Lyon, Ecully, France, KAI SCHNEIDER, M2P2-CNRS & CMI Aix-Marseille University, Marseille, France — The mixing of a passive scalar blob in a confined vessel is studied. A flow is generated by a rod, describing a figure-eight motion. The two-dimensional incompressible Navier-Stokes and advection-diffusion equations are solved using direct numerical simulation with no-slip and no-flux boundary conditions for the velocity and scalar, respectively. These boundary conditions are imposed on the wall and the rod by using a volume penalization method as described in in Ref. [1], in combination with a classical Fourier pseudo-spectral code. The decay of scalar variance in Stokes regime, for different Schmidt numbers, is compared with the one obtained in Ref. [2] for chaotic mixing. Subsequently, the influence of Reynolds and Schmidt numbers on turbulent mixing is investigated. In order to quantify the mixing at infinite Schmidt number, we measure the dispersion of tracer particles. Both the variance and higher moment statistics for the scalar concentration are analyzed. We show that the scalar variance decays in time following a powerlaw.

[1] B. Kadoch, D. Kolomenskiy, K. Schneider and P. Angot. J. Comput. Phys., 231, 4365-4383, 2012.
[2] E. Gouillart, O. Dauchot, B. Dubrulle, S S. Roux, and J.-L. Thiffeault. Phys. Rev. E 78, 026211, 2008.

Sunday, November 18, 2012 4:45PM - 5:50PM –

Session E24 Acoustics III 30E - Chair: Shreyas Mandre, Brown University

4:45PM E24.00001 An Experimental Study of a Nonlinear Acoustic Lens Interfaced with Water , CARLY DONAHUE, PAUL ANZEL, THOMAS KELLER, CHIARA DARAIO, California Institute of Technology, Graduate Aerospace Laboratories — Acoustic waves are routinely used in imaging and excitation applications such as in ultrasonic imaging or hyperthermia surgery. However, current acoustic technology is limited by focal resolution and maximum amplitude. In this work, we have constructed a nonlinear acoustic lens, which is composed of an array of chains of steel spherical particles supported by a matrix. The nonlinearity of the system originates from the contact interaction between the particles, which enables the formation of solitary waves in the chains. The acoustic lens can be designed and interfaced with a target medium such that when the solitary waves exit the chains, the waves coalesce at a focal point. The highly compact acoustic waves at the focus are called "sound bullets." Additionally, since the solitary wave speed increases as the pre-compression between the spheres increases, the focal point can be controlled mechanically. In this work, we use water as our target medium. Measurements are taken using a hydrophone that is scanned over an area to produce a two dimensional pressure map. The chains are separated from the water using cover plates, the choice of which strongly influences the transmission of the solitary wave into the host medium.

4:58PM E24.00002 Multiple Scales Analysis of a Thermoacoustic Heat Pump , MICHAEL MILLER, SHREYAS MANDRE, Brown University — Thermoacoustics utilizes the temperature and density oscillations inherent to acoustic vibrations coupled with heat conduction near a wall to produce heat transfer from sound (or sound from a heat source). In the heat pump setup, thermal energy is transferred to the wall from an element of gas during compression and taken from the wall during rarefaction. In thermoacoustic phenomena, acoustic oscillations occur on a very short time scale while heat transfer occurs over many acoustic cycles. Therefore, multiple scales analysis is well suited to describe the physics. We present a multiple scales analysis for a narrow two-dimensional channel between two thin, non-stationary plates resulting in an integral equation for the temperature distribution along the channel as a function of the long time scale. We solved this equation numerically to find a steady state solution for a given set of parameters.

5:11PM E24.00003 Improved parabolization of the compressible Euler equations , AARON TOWNE, TIM COLONIUS, Caltech — 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 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 to explicitly remove the upstream acoustic mode from the linearized Euler equations. The eigenvalue associated with the upstream mode is zeroed in Fourier-Laplace space, resulting in a non-local propagation equation after the transforms are inverted. The non-locality arises from terms involving the square root of the Fourier-Laplace variables. In order to recover local, real-space PDEs from the transformed equations, these terms are approximated using Pade-type rational approximation. The resulting equations are parabolic in the downstream direction and require no damping for a stable march. We will outline this method and present results that demonstrate its stability and accuracy for different approximations of the square root.

5:24PM E24.00004 Experimental evaluation of sound produced by two cylinders in a cross flow in various configurations , MICHAEL BILKA, PETER KERRIAN, SCOTT MORRIS, University of Notre Dame — Cylinders in a cross flow is a canonical test case that is considered representative of industrial fluid flow problems, such as heat exchangers and aircraft landing gear. The general configurations are cylinders in tandem, parallel or staggered, which lead to several interesting flow phenomena such as wake cavity/wake interaction effects (tandem), symmetric/asymmetric wake behavior (parallel) or wake interaction/coanda effects (staggered), depending on relative location of the cylinders. In many cases, it is important to not only understand the flow interaction between the cylinders but also the acoustic consequence of such configurations. However, information on the acoustical behavior based on these configurations is relatively small compared to that of the steady aerodynamic and flow interaction behavior. The present work investigates the acoustic spectral properties of these configurations in order to characterize the sound produced by cylinder proximity and wake effects. The spectra are measured using an acoustic beam-forming technique to identify the sources and remove spurious content from the spectra.

5:37PM E24.00005 Sensitivity of meteor infrasound to atmospheric uncertainties , CHRISTOPHE MILLET, CEA, DAM, DIF, F-91297 Arpajon, France, CHRISTOPHE HAYNES, LPTMC, Université Paris 6, 75252 Paris, France — In recent years, numerous bolide sources have been detected by the IMS infrasound arrays. Even though a variety of waveform data may be extracted from recorded signals, only a few parameters are used throughout meteor research, the most common being the arrival time of a signal. Other data forms include the amplitude and duration of the signal. As the shock wave depends on the properties of the medium in which it travels, a full analysis of the atmosphere for any event is required. In the present work, we model the propagation of a shock wave through a randomly layered atmosphere. In a deterministic or random environment, a generated meteoric shock wave propagates from a strong blast region out to the far-field acoustic limit. Inclusion of a random atmosphere will then affect all possible outcomes of the ray path. The resulting amplitude and period of any N-wave signal at ground level are obtained using both a ray tracing method and a theoretical approach based on Whitham's method. This method becomes particularly relevant when applied to the crater-forming meteorite fall near Carancas, Peru (2007); given that the specific trajectory of the meteor was unknown and that the maximum amplitude of the recorded signal were substantially affected by atmospheric conditions.

Sunday, November 18, 2012 4:45PM - 5:50PM –

Session E25 Flow Control: Jets 31A - Chair: Dan Henningson, KTH

4:45PM E25.00001 Modeling Forced Jets with Parabolized Stability Equations , ANIRUDDHA SINHA, TIM COLONIUS, California Institute of Technology — The theory of parabolized stability equations (PSE) is employed to model the forcing response of high-speed and high Reynolds number round jets. Recently, this theory has been successfully applied to unforced subsonic and supersonic jets to predict the dynamics of the wavepackets implicated in noise radiation to aft angles. The effect of some unsteady actuation mechanisms (e.g. those based on plasma, fluidic injection, etc.) may be practically considered to be impulsive in nature rather than sinusoidal. As a first approximation, linear PSE modes, computed as instabilities of the unforced jet, are suitably combined to simulate the impulse response. The predictions from this procedure compare reasonably well with experimentally determined phase-averaged near-field pressure in subsonic jets forced with localized arc filament plasma actuators. The model can be improved by incorporating the nonlinear interactions between the excited modes in the jet. Such a model is being applied to predict the changes to the dynamics of a supersonic jet actuated by pulsed air injection. The end goal is to use the validated PSE model to understand noise source mechanisms, and propose novel actuation strategies for their mitigation.

4:58PM E25.00002 Effect of Orifice Angle on Synthetic Jet Actuators , POOYA KABIRI, KEVTA Fire Systems, DOUG BOHL, Clarkson University — Synthetic Jet Actuators (SJA's) are commonly used in flow control experiments. It has been shown that in many applications the direction of the jet with respect to the flow direction and surface geometry is important. For example, when controlling separation on an airfoil, the maximum control is obtained when the exit jet is directed parallel, rather than perpendicular to the surface. Typically, the SJA itself must be rotated to direct the flow. In some cases this may not be possible and the orifice must be oriented to direct the flow. This work investigates the flow field induced by a SJA's with the exit orifice located on the surface opposite of the membrane and differing exit angles. The orifices investigated are oriented straight (i.e. 90˚) or angled (60˚, 30˚) with respect to the membrane surface. The entrances to the orifices are contoured to inhibit flow separation within the orifice. The results show that the peak exit velocity is found when the orifice is straight and systematically reduces as the angle becomes smaller. The results also show that the external flow field is greatly affected by the orifice angle. When the orifice is straight a pair of vortices are formed that convect away from the slot. As the angle is reduced a boundary layer is formed on the external surface of the SJA. This boundary layer changes the formation of the vortex pair and inhibits convection of the vortices away from the slot.

5:11PM E25.00003 Spatio-Temporal Flow Structure over a NACA-0015 Airfoil under the influence of ZNMF Jet Flow Control , CALLUM ATKINSON, NICOLAS BUCHMANN, JULIO SORIA, Monash University — The spatio-temporal flow structure associated with Zero Net Mass Flux (ZNMF) jet forcing at the leading edge of a NACA-0015 airfoil (Re = 30000, AoA = 18 deg) is investigated using high-repetition rate Particle Image Velocimetry (HR-PIV). In the absence of forcing, flow separation occurs at the leading edge, while ZNMF jet forcing at a frequency of f+ = 1.3 leads to a nearly complete reattachment with a 45% lift increase. The structure and dynamics associated with both the forced and unforced case are considered and the dominant frequencies are identified. A triple-decomposition of the velocity field is performed to identify the spatio-temporal perturbations produced by the ZNMF jet forcing. This forcing results in a reattachment of the flow, which is caused by the generation of large-scale vortices that entrain high momentum fluid from the freestream. Forcing produces a series of vortices that are advected parallel to the airfoil surface. Potential mechanisms by which these vortices affect the flow reattachment are discussed.

5:24PM E25.00004 Using a wall-normal jet to modify the large-scale structures in a turbulent boundary layer , MURALI KRISHNA TALLURU, BRETT BISHOP, NICHOLAS HUTCHINS, CHRIS MANZIE, IVAN MARUSIC, The University of Melbourne, THE UNIVERSITY OF MELBOURNE TEAM — We report on attempts to use a wall-normal jet to modify the large-scale structures ("super structures") that are known to populate the logarithmic regions of high Reynolds number turbulent boundary layers. An upstream spanwise array of surface mounted shear-stress sensors detects the passage of the large-scale events. A rectangular wall-normal jet, located downstream of this array targets the identified event and a second spanwise array downstream of the jet monitors any alterations to the large-scale structure. A traversing hot wire probe is mounted above the downstream array to look for modifications across the depth of the boundary layer. As a first step, an off-line control strategy is investigated. In this case, there is no active controller, the jet is periodically fired with fixed parameters and during post-processing, the "control" strategy is emulated in a conditional sense to understand the interactions of an actuated jet with the larger turbulent structures. The results from off-line control scheme are used to develop a real-time control scheme to systematically target the large-scale high skin friction events. The outcome of this control approach on both the instantaneous coherent structures and also the time-averaged quantities is investigated.

5:37PM E25.00005 Synthetic Jet Control of a Yawing Axisymmetric Body[1] , THOMAS LAMBERT, BOJAN VUKASINOVIC, ARI GLEZER, Georgia Institute of Technology — The global aerodynamic forces and moments on an axisymmetric yawing body are controlled in wind tunnel experiments by exploiting the interaction of an array of synthetic jet actuators with the cross flow over the tail section of the body. The model is supported by a vertical wire through its aerodynamic center and is free to move in yaw. The baseline motion of the model is a yaw oscillation with amplitude and frequency that both monotonously increase with free stream velocity, characteristic of vortex shedding. The aft-facing control jet actuators emanate from narrow, azimuthally segmented slots around the perimeter of the tail section, and activation of the control jets effects the model's path through localized flow attachment on integrated Coanda surfaces. The control jets are used to control the yaw trajectory of the model using a closed loop PID controller. The baseline and controlled model motion is monitored using a laser vibrometer, and the flow evolution near the body and in its near wake is investigated using PIV. The coupled, time dependent response of the model to the actuation is investigated with emphasis on controlling its unstable modes.

[1]Supported by ARO

Sunday, November 18, 2012 4:45PM - 5:50PM –

Session E26 Reactive Flows III: Supersonic Combustion and Theory 31B - Chair: Heinz Pitsch, Stanford University

4:45PM E26.00001 *A priori* and *a posteriori* analyses of the flamelet/progress variable approach for supersonic combustion , AMIRREZA SAGHAFIAN, HEINZ PITSCH, Stanford University — A compressible flamelet/progress variable approach (CFPV) has been devised for high-speed flows. Temperature is computed from the transported total energy and tabulated species mass fractions and the source term of the progress variable is rescaled with pressure and temperature. The combustion is thus modeled by three additional scalar equations and a chemistry table that is computed in a pre-processing step. Three-dimensional direct numerical simulation (DNS) databases of reacting supersonic turbulent mixing layer with detailed chemistry are analyzed to assess the underlying assumptions of CFPV. Large eddy simulations (LES) of the same configuration using the CFPV method have been performed and compared with the DNS results. The LES computations are based on the presumed subgrid PDFs of mixture fraction and progress variable, beta function and delta function respectively, which are assessed using DNS databases. The flamelet equation budget is also computed to verify the validity of CFPV method for high-speed flows.

4:58PM E26.00002 The bifurcation of scramjet unstart[1] , IK JANG, JOSEPH NICHOLS, PARVIZ MOIN, Stanford University — The bifurcation of scramjet unstart is numerically investigated. Near an unstart event, a small change made to the heat released in the combustor causes catastrophic changes in the system. In this talk, this bifurcation structure is studied not only for simple idealized flows but also for a more realistic multi-dimensional model scramjet. In the hysteresis zone of the bifurcation structure, steady but unstable solutions are calculated by means of pseudo-arclength continuation. Pseudo-arclength continuation is performed using Newton-Raphson iteration based on Jacobians computed using an automatic differentiation technique. In addition, eigendecomposition is performed to extract the least stable eigenfunctions describing the system dynamics. Finally, the unstart mechanism and the margins to unstart are evaluated from the bifurcation structure and the linearized system dynamics.

[1]Supported by the PSAAP program of DOE

5:11PM E26.00003 Quantification of multiple types of uncertainties in the HyShot II scramjet , JOHAN LARSSON, MICHAEL EMORY, PAUL CONSTANTINE, NICOLAS KSEIB, JAVIER URZAY, FRANCISCO PALACIOS, CATHERINE GORLE, GIANLUCA IACCARINO, Stanford University — The talk describes a collaborative effort at quantifying the effects of different types of uncertainties on the pressure rise in the HyShot II scramjet combustor. These uncertainties include the shock-tube conditions (mean and turbulent state, angle-of-attack), the chemical reaction rates in the hydrogen-air chemistry, and errors in the assumed stress-strain relationship in the RANS turbulence model. A physics-based strategy for dimensional reduction is used to make the study computationally feasible. Results include the ranking of different types of uncertainties.

5:24PM E26.00004 Flame front as hydrodynamic discontinuity[1] , YASUHIDE FUKUMOTO, Kyushu University, Japan, SNEZHANA ABARZHI, University of Chicago, Chicago, IL, USA — We applied generalized Rankine-Hugoniot conditions to study the dynamics of unsteady and curved fronts as a hydrodynamic discontinuity. It is shown that the front is unstable and Landau-Darrieus instability develops only if three conditions are satisfied (1) large-scale vorticity is generated in the fluid bulk; (2) energy flux across the front is imbalanced; (3) the energy imbalance is large. The structure of the solution is studied in details. Flows with and without gravity and thermal diffusion are analyzed. Stabilization mechanisms are identified.

[1]NSF 1004330

5:37PM E26.00005 The effect of the Darrieus-Landau instability on statistically planar turbulent flames , NAVIN FOGLA, MOSHE MATALON, University of Illinois at Urbana Champaign, FRANCESCO CRETA, University of Rome "La Sapienza" — In a recent work, the propagation of premixed flames in weak two dimensional homogeneous turbulent flows was studied by Creta & Matalon using a hybrid Navier-Stokes/interface capturing technique within the context of a hydrodynamic model, which treats the flame as a surface of density discontinuity. Particular attention was devoted to the Darrieus-Landau (DL) instability, whose effect on turbulent flames has recently been recognized both via experiments and numerical studies. This instability can be triggered in a laboratory setting via changes in pressure and/or fuel type and composition. In the current study, we focus on the effects of the DL instability on turbulent, statistically planar flames. Results are therefore limited to positive Markstein length corresponding to lean hydrocarbon-air or rich hydrogen-air mixtures. We show that conditions under which a laminar flame remains unaffected by the instability, a turbulent flame on the other hand is affected by it. The turbulent flame is observed to exhibit, in addition to the effect of thermal expansion, a nontrivial dependence on the instability, and on the turbulent integral scale both effects modulating, the well established quadratic dependence of turbulent flame speed on turbulence intensity.

Sunday, November 18, 2012 4:45PM - 5:50PM –

Session E27 Renewable Energy 31C - Chair: Mehdi Raessi, University of Massachusetts, Dartmouth

4:45PM E27.00001 Energy harvesting of fluttering piezoelectric flags , OLIVIER DOARE, UME, ENSTA-Paristech, SEBASTIEN MICHELIN, LadHyX, Ecole Polytechnique, JIAWAN CHEN, UME, ENSTA-Paristech, YIFAN XIA, LadHyX, Ecole Polytechnique — The energy harvesting from a flutter instability of a plate equipped with adjacent pairs of piezoelectric elements shunted with independent resistive circuits is considered. When the length of the piezoelectric elements is low compared to the typical wavelengths of bending deformations, governing equations are derived in the form of continuous coupled fluid-solid-electrical equations. These equations are used to perform a linear stability analysis of the coupled system, and a parametric study of the efficiency of the energy transfer from the fluid-solid system to the electrical system addressing both linear and nonlinear dynamics.

4:58PM E27.00002 Flexible Beams in Turbulent Boundary Layers for Piezoelectric Energy Harvesting[1] , HUSEYIN DOGUS AKAYDIN, NIELL ELVIN, YIANNIS ANDREOPOULOS, The City College of The City University of New York — Thin flexible cantilever beams with patches of piezoelectric materials or surrogates with strain gages attached have been placed inside turbulent boundary layers to search for the maximum energy output. A turbulent boundary layer (TBL) carries mechanical energy distributed over a range of temporal and spatial scales and their interaction with the immersed piezoelectric beams results in a strain field which generates the electrical charge. This energy harvesting method can be used for developing self-powered flow sensors. In the present experimental work TBLs with Re_θ between 1500 and 7700 were configured in a large scale wind tunnel. The orientation of the beam relative to the incoming flow and its distance to the wall was found to be critical parameters affecting the energy output. "Power maps" generated by testing a beam in TBLs at different free stream velocities and wall distances will be presented. Vibration amplitudes and frequencies at several principal orientations will be compared. The effect of yaw angle, pitch angle, length and natural frequency of the beam will be examined. The role of instantaneous pressure fluctuations and large-scale TBL structures in this rather complex fluid-structure interaction will be discussed in interpreting the electrical output results.

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

5:11PM E27.00003 On the effects of energetic coherent motions on power and wake dynamics of an axial-flow marine turbine in an open channel[1] , CRAIG HILL, LEONARDO CHAMORRO, University of Minnesota, VINCENT NEARY, BUDI GUNAWAN, Oak Ridge National Laboratory, ROGER ARNDT, FOTIS SOTIROPOULOS, University of Minnesota, U. OF MN - SAFL TEAM, ORNL COLLABORATION — Laboratory experiments are carried out to study the effect of energetic coherent motions on the performance of a model axial-flow hydrokinetic turbine in the main channel at Saint Anthony Falls Laboratory. A mechanical system allows the turbine angular velocity to be precisely specified and maintained via a controller. Power fluctuations are tracked using a torque sensor connected to the turbine structure. Periodic energetic coherent motions are introduced in the flow by placing a cylinder at various locations upstream of the turbine on the plane of symmetry. Three acoustic Doppler velocimeters and a torque sensor are used to obtain synchronous high resolution measurements of the flow and turbine power at a rate of 200 Hz, respectively. Flow measurements are obtained at various locations upstream and downstream of the turbine. The measurements provide novel insights into the role of strong energetic coherent motions on turbine power fluctuations, tip vortex instability, and mean wake recovery. The implications of these findings for the efficient operation of hydrokinetic turbines in natural waterways will be discussed.

[1]Support and funding provided by Oak Ridge National Laboratory and the Office of Energy Efficiency and Renewable Energy of the Department of Energy under DOE Contract DE-AC05-00OR22725.

5:24PM E27.00004 Computational simulation of ocean wave energy converters using the fast fictitious domain method , AMIRMAHDI GHASEMI, ASHISH PATHAK, MEHDI RAESSI, University of Massachusetts Dartmouth — Ocean wave energy is considered one of the major renewable energy resources. We are developing a computational tool for analysis of wave energy converters. This computational tool is envisioned to complement and leverage experimental knowledge base, which is expensive or difficult to develop in this field. The computational tool simulates the interaction of two-phase fluid flows with a moving solid object by solving the full Navier-Stokes equations. Unlike previous models, it considers all non-linear effects, e.g. wave breaking and fluid-solid interactions. We use the two-step projection method in the finite-volume context with GPU acceleration to solve the flow equations. The fluid interfaces are tracked by using the volume-of-fluid (VOF) method. We incorporated the fast fictitious-domain method into our flow solver to simulate the interactions of a moving solid object with two-phase flows. Extending Youngs' piecewise linear interface reconstruction technique, we use a geometrical reconstruction of liquid-gas-solid interfaces at the triple point to accurately track the three phases. We will present results of canonical test cases, which demonstrate the accuracy of the above approach, as well as a 2D simulation of a buoy interacting with water waves in a tank.

5:37PM E27.00005 An experimental study of small-scale flexible wind turbine blades[1] , PARIYA POURAZARM, YAHYA MODARRES-SADEGHI, MATTHEW LACKNER, University of Massachusetts — With the increasing size of offshore wind turbine rotors, the design criteria used for the blades may also evolve. Current offshore technology utilizes three relatively stiff blades in an upwind configuration. With the goal of minimizing mass, there is an interest in lightweight rotors that instead utilize two flexible blades oriented downwind. These design possibilities necessitate a better understating of the fundamental behavior of such flexible blades. In the current work, a series of experiments are conducted using a small scale wind turbine built with adjustable features. The blades are designed using relatively thin, low Reynolds number airfoils and built using rapid-prototyping methods with a flexible material. The number of blades as well as their pitch angle, stiffness, and distance from the tower can be varied. The tests are conducted in a wind tunnel with a cross-section of 1 m by 1 m, a wind speed range of 3 to 20 m/s and a turbulence intensity of less than 1%. The small scale wind turbine is tested both upwind and downwind and a dynamic strain gauge is placed on the blades to measure blade deflection and dynamic loading in various configurations.

[1]The support provided by the Wind Technology Testing Center, a part of the Massachusetts Clean Energy Center is acknowledged.

Sunday, November 18, 2012 4:45PM - 5:50PM –

Session E28 Free-Surface Flows III 32A - Chair: Neil Ribe, Laboratoire FAST, Orsay

4:45PM E28.00001 Bound-state formation in falling liquid films[1] , PHUC-KHANH NGUYEN, Department of Mechanical Engineering, University of Thessaly, MARC PRADAS, SERAFIM KALLIADASIS, Department of Chemical Engineering, Imperial College London, VASILIS BONTOZOGLOU, Department of Mechanical Engineering, University of Thessaly — Direct numerical simulation shows that the interaction between solitary pulses may give rise to the formation of bound states consisting of two or more pulses separated by well-defined distances and traveling at the same velocity. Stationary pulse couples are studied first. The resulting equilibrium pulse distances compare favorably to theoretical predictions at large and intermediate pulse separations. When the two pulses are closely spaced, the theory becomes increasingly less accurate. Their time-dependent simulations indicate that all initial conditions of large separations lead to a monotonic attraction or repulsion to the stable bound states. However, intermediate range leads to a self-sustained oscillatory variation of the pulse separation distance, with well-defined amplitude and period, and a mean separation coinciding with the stationary distance. Eventually a very close separation causes an explosive repulsion of two pulses toward much larger stable separation. Bound states consisting of three pulses are computed next. The equilibrium separation distances in a symmetric system are similar to predictions based on simple couples. However, in an asymmetric one, they deviate significantly from simple predictions.

[1]Partially supported by FP7-Marie Curie ITN-"MULTIFLOW"-GA-214919-2

4:58PM E28.00002 Deformation of a Thin Film by a Wall Jet , NAIMA HAMMOUD, TALAL AL-HOUSSEINY, HOWARD STONE, Princeton University, Princeton, NJ — A variety of industrial processes such as jet stripping or jet wiping involve a high speed stream of gas flowing over a liquid film. In this work, we model this kind of situation by considering a thin viscous liquid film, over which a high Reynolds number laminar wall jet (or Glauert jet) is flowing. We study the shape of the thin liquid film, which is deformed due to the shear stress induced by a jet of a low-viscosity fluid. The mechanics of the jet, which is modeled by boundary-layer theory, is coupled to the mechanics of the thin film, which includes the influence of surface tension and buoyancy. We describe the unsteady shape of the film using the lubrication description to derive a nonlinear PDE that is coupled to the Glauert jet via interfacial stresses. For the steady state, we obtain analytical solutions in different asymptotic regimes. We compare our theoretical findings to numerical simulations conducted with the finite volume solver FLUENT.

5:11PM E28.00003 High spatio-temporal resolution PIV of laminar boundary layer relaxation instability at the free surface of a jet[1] , MATTHIEU ANDRE , PHILIPPE BARDET, The George Washington University — In high-speed free surface flows, microscale instabilities can lead to dramatic macroscale effects such as waves, breakup, or air entrainment. The importance of jets in practical applications requires a better understanding of the mechanisms leading to these instabilities. This experimental study focuses on laminar boundary layer relaxation (LBLR) instability. This has received fewer attention than other instabilities due to the small scale, the high Reynolds number and the proximity of an interface. The experiment features a $20.3mm \times 146.0mm$ laminar slab wall jet exiting a nozzle into quiescent air (Re=3.1×10^4 to 1.6×10^5). The free surface is flat near the nozzle exit then the LBLR leads to 2D capillary waves which can become very steep eventually resulting in primary breakup and air entrainment. The inception and growth of the capillaries are investigated using time-resolved PIV coupled with PLIF to track the free surface. A magnification of 4 allows a spatial and temporal resolution better than 0.1mm and 0.1ms, respectively. These high resolution results show the role of vortices -created by the roll-up of the shear layer below the surface- in the formation of capillaries. Vortices and waves are a coupled system; the waves can sustain, damp, or amplify.

[1]This study has been supported by the start-up funds from The George Washington University to Dr. Bardet.

5:24PM E28.00004 Torricelli's curtain: morphology of horizontal laminar jets under gravity , NEIL RIBE, GABRIEL PATERNOSTER, MARC RABAUD, Laboratoire FAST, Orsay — It has been "known" since the seventeenth century that a jet of water issuing horizontally from a hole in the side of a bucket describes a parabolic trajectory. However, this bit of canonical fluid mechanical lore is wrong in many cases. Our recent experiments performed on laminar jets issuing from a horizontal tube show that the initially round jet typically evolves into a thin vertical curtain bounded by bulbous rims at its upper and lower extremities. Moreover, injected dye reveals the presence of a recirculating flow with helical streamlines around the jet's axis. To understand this behavior, we formulate an analytical model for the near-orifice structure of the jet in the limit of large Froude number $Fr \equiv \epsilon^{-1} \gg 1$. We find that a recirculating flow is generated by the sinusoidal variation of the nonhydrostatic pressure around cross-sections of the jet at order ϵ, and that deformation of the cross-section occurs at order ϵ^2. We also use the volume-of-fluid code Gerris to study numerically the evolution of the jet's morphology as a function of the Reynolds, Froude and Ohnesorge numbers, and compare the results with our analytical theory and with laboratory experiments.

5:37PM E28.00005 Spatial Coherence Resonance for Maximizing Self-Organization and Pattern Fidelity in Free Surface Films , SANDRA TROIAN, NAN LIU, California Institute of Technology, MC 128-95, Pasadena, CA 91125 — The influence of external modulation on pattern formation can offer significant insight into hydrodynamic behavior and provide an alternate means of optimizing the self-organization process. While temporal modulation has been used to great effect for decades, there has been less emphasis on external spatial forcing as a way of enhancing pattern uniformity. The majority of studies investigated in the context of spatial coherence phenomena have involved systems undergoing pattern formation in 2D. In this talk, we call attention to hydrodynamic instabilities in thin films in which microarray structures emerge spontaneously in 3D with no intrinsic steady states unless film depletion occurs. However, these structures are highly prone to defects and difficult to control over large areas. For this talk, we focus on an example involving the deformation of a free surface nanofilm exposed to a large transverse thermal gradient whose magnitude is spatially modulated near the resonance point. By a combination of weakly nonlinear analysis and numerical simulations, we demonstrate the existence of a spatial coherence regime leading to rapid growth of perfectly uniform microarrays with high pattern fidelity and even denser packing than possible without spatial forcing.

Sunday, November 18, 2012 4:45PM - 5:50PM –

Session E29 Porous Media Flows III 32B - Chair: Marcus Roper, University of California, Los Angeles

4:45PM E29.00001 Film formation in a vertical Hele-Shaw cell , THOMAS WARD, Iowa State University, Department of Aerospace Engineering, ERIC FINLEY, North Carolina State University, DEON WILKINS, Bethune-Cookman University, MICHAEL SULLIVAN, North Carolina State University — A Hele-Shaw cell containing silicone oil is initially displaced in the vertical direction using electrical forces and capillary pressure. Once the equilibrium height is reached, electrical actuation is halted and the system moves to a new equilibrium height based on capillary and hydrostatic forces. Here, the focus is on film formation from the oil to the boundary walls of the Hele-Shaw cell. A theoretical model is used to predict silicone oil/air interface speed. The theory is also compared with experimental data. The resulting differences between predicted equilibrium heights based on theory and actual experimental equilibrium heights are used to predict the thickness of any resulting oil film on the walls which increases the capillary pressure. Plate separation distances of 300, 500, and 750 micrometers are used with silicone oil viscosities of 10, 100 and 1000 cSt to determine the effect of these parameters on film formation. A variety of initial heights are also used to determine how the initial rise height affects film formation as well. By varying these parameters, a wide range of Reynolds and capillary numbers are examined and their corresponding effect on film formation is determined.

4:58PM E29.00002 Pattern formation in poroelastic systems , CHRISTOPHER MACMINN, JOHN WETTLAUFER, ERIC DUFRESNE, Yale University — Poroelastic effects, where fluid flow through a porous solid is coupled to elastic deformation of the solid, play an important role in many natural and engineering systems. Due to the highly nonlinear nature of the fluid-solid coupling in these systems, instability-driven pattern formation is both likely and very poorly understood. Here, we use laboratory experiments to explore pattern formation in a model poroelastic system. We study the paradigmatic problem of fluid injection into a quasi-two-dimensional porous medium, and we show that poroelasticity results in a nonlocal coupling between the fluid and the solid that drives pattern formation in even relatively simple fluid flows.

5:11PM E29.00003 Inertial flow on micropatterned surfaces: Modeling polygonal water bells , EMILIE DRESSAIRE, Trinity College, LAURENT COURBIN, Institut de Physique de Rennes, CNRS, France, ADRIAN DELANCY, Harvard University, MARCUS ROPER, UCLA, HOWARD STONE, Princeton University — Regularly micropatterned substrates are commonly used to study complex phenomena such as spreading and splashing. We have used a well characterized hydrodynamic object, a water bell to characterize the flow on such rough surfaces. In the water bell configuration, the thin liquid film and the solid surface interact over a short lengthscale and viscous effects are negligible. We develop a simple model that shows the role of hydrodynamic interactions in the networks of microposts. We are able to predict the shape of the polygonal water bells. By considering the flow in the rim, we also identify preferred sites of droplet emissions.

5:24PM E29.00004 Universality Results for Multi-Layer Hele-Shaw and Porous Media Flows[1] , PRABIR DARIPA, Texas A&M University — Saffman-Taylor instability is a well known viscosity driven instability of an interface. Motivated by a need to understand the effect of various injection policies currently in practice for chemical enhanced oil recovery, we study linear stability of displacement processes in a Hele-Shaw cell involving injection of an arbitrary number of immiscible fluid phases in succession. This is a problem involving many interfaces. Universal stability results have been obtained for this multi-layer (multi-region) flow in the sense that the results hold with arbitrary number of interfaces. These stability results have been applied to design injection policies that are considerably less unstable than the pure Saffman-Taylor case. In particular, we determine specific values of the viscosity of the fluid layers corresponding to smallest unstable band. Moreover, we discuss universal selection principle of optimal viscous profiles. The talk is based on following papers.
P. Daripa and X. Ding, "Universal Stability Properties for Multi-Layer Hele-Shaw Flows and Application to Instability Control," SIAM J. Appl. Math., To Appear.
P. Daripa and X. Ding, "Selection principle of optimal profilesfor Multi-Layer flows," Trans. Porous Media, To Appear.

[1]Qatar National Fund (a member of the Qatar Foundation)

5:37PM E29.00005 ABSTRACT WITHDRAWN –

Sunday, November 18, 2012 4:45PM - 6:03PM –

Session E30 Nanofluids: Computations II 33A - Chair: Jeremy Templeton, Sandia National Laboratories

4:45PM E30.00001 Fluid flows in nano/micro network configurations: a multiscale molecular-continuum approach[1] , MATTHEW BORG, University of Strathclyde, DUNCAN LOCKERBY, University of Warwick, JASON REESE, University of Strathclyde — We present a new hybrid molecular-continuum methodology for resolving multiscale flows emergent in nano-/micro-scale networks, in particular for NEMS/MEMS applications. The method models junction and channel components of the network using independent MD micro elements. Long channels with uniform or gradually varying nano-scale sections along the direction of flow, contribute the most towards the highest computational savings, by replacing them with much smaller MD simulations. Junction components, however, do not exhibit any length-scale separation and are modelled in their entirety. All micro elements are coupled together in one hybrid simulation using standard continuum fluid-dynamics equations, that dictate the overall macroscopic flow in the network. In the case of isothermal, incompressible, low-speed flows we use the conservative continuity and momentum equations. An iterative algorithm is presented that computes at each iteration the new constraints on the pressure differences applied to individual micro elements, in addition to enforcing overall continuity in the network. We show that the hybrid simulation of various small network cases converge quickly to the result of a full MD simulation over just a few iterations, with significant computational savings.

[1]This work is financially supported by the EPSRC Programme Grant EP/I011927/1.

4:58PM E30.00002 Realistic molecular dynamics simulations of water flow through carbon nanotube membranes , JASON REESE, WILLIAM NICHOLLS, MATTHEW BORG, Mechanical & Aerospace Engineering, University of Strathclyde, Glasgow G1 1XJ, UK, DUNCAN LOCKERBY, School of Engineering, University of Warwick, Coventry CV4 7AL, UK, MICRO AND NANO FLOWS FOR ENGINEERING COLLABORATION[1] — Water transport through (7,7) carbon nanotubes (CNTs) is investigated using non-equilibrium molecular dynamics simulations. In particular, we are interested in how the CNT length and the prevalence of defects affects the internal flow dynamics. Pressure-driven water flow through CNTs ranging from 2.5 nm to 50 nm long is simulated. Structural defects are modeled as vacancy sites (missing carbon atoms). We demonstrate that under the same applied pressure difference an increase in CNT length has a negligible effect on the resulting mass flow rate and fluid flow velocity. Flow enhancements over hydrodynamic expectations are in fact directly proportional to the CNT length. Axial profiles of fluid properties demonstrate that CNT inlet and outlet effects are significant: large viscous losses in these regions contrast with central "developed flow" regions in longer CNTs where the flow is almost frictionless. Increasing the numbers of defects in the CNT structure does, however, lead to significant reductions in the fluid velocity and mass flow rate through the CNTs.

[1]http://www.micronanoflows.ac.uk/

5:11PM E30.00003 Combined Temperature/Momentum Boundary Conditions for Molecular Dynamics Simulations of Flow in Nanofluidic Systems[1] , JEREMY TEMPLETON, REESE JONES, Sandia National Laboratories, California — Molecular dynamics (MD) is a useful technique for scientific investigations of nanofluidic processes as it explicitly represents the dynamics of every atom in a system. In order to model systems of interest, e.g. nanochannels, it is necessary to constrain atomic motions to conform to conditions corresponding to large-scale information, e.g. thermodynamic variables. However, many engineered configurations involve complex interactions between the system and its environment, and take place in non-trivial geometries. To accurately simulate these phenomena, methods to apply boundary conditions to MD systems are required that simultaneously regulate the temperature (i.e., energy) and momentum of the atoms in a local manner. This work uses an atomistic-to-continuum formulation to generate boundary conditions by using finite elements (FE) and their associated shape functions to define "boundaries" for a particle system. By projecting onto the FE basis, coarse-scale observables are identified for regulation based on separating the mean and fluctuating velocity components defining the momentum and temperature. Regulating the MD system is achieved by applying constraints posed on the coarse-grained variables. The method is illustrated by application to several nanofluidic systems.

[1]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

5:24PM E30.00004 Comparing Molecular Dynamics Models for Electrolyte Solutions in Nanochannels[1] , JONATHAN LEE, JEREMY TEMPLETON, Sandia National Laboratories — In electrolyte modelling, it is common to simplify the solvent using the three-component model (3CM), i.e. a single-site, chargeless Lennard-Jones atom as the solvent component. To account for the dielectric nature of typical solvents, a relative permittivity value is applied to all Coulombic interactions, thus weakening ion-ion interactions as if each ion is surrounded by a solvation shell. Fluid Density Functional Theory, Monte Carlo simulation, and molecular dynamics (MD) simulation all commonly employ the 3CM to facilitate calculations, but the consequences are not well characterized. We used MD to compare the 3CM electrolyte to a molecular solvent model (MSM) where the solvent is a three-site H_2O) molecule. Special care was taken to compare cases with the same thermodynamic state by having a quantifiable reference state, and cases covered a range of applied surface charge in a nanochannel configuration. At a glance, the two models give qualitatively similar density profiles. However, we find that many profile features, physical quantities such as electric field and potential, as well as ionic packing structure near the surface evolve quite differently as the load is varied.

[1]Sandia National Laboratories is a multi-program laboratory managed and operated by Sandia Corporation, a wholly owned subsidiary of Lockheed Martin Corporation, for the USDoE's National Nuclear Security Administration under contract DE-AC04-94AL85000.

5:37PM E30.00005 Bouncing, splashing and disintegrating nanodrops , JOEL KOPLIK, RUI ZHANG, City College of CUNY — The impact of nanometer-sized drops on solid surfaces is studied by molecular dynamics simulations. The surfaces are atomically smooth, dry and non-wetting, and both volatile and non-volatile liquids are considered. At low impact velocities drops distort on contact but bounce off the surface and relax back to a spherical shape. At higher velocities drops form a prompt splash on impact and subsequently disintegrate, while at still higher velocities drops disintegrate immediately on impact. In contrast to macroscopic drops, the presence or absence of vapor plays no role at all in nanodrop splashing.

5:50PM E30.00006 Nanodrop impact on rough and textured surfaces , RUI ZHANG, JOEL KOPLIK, City College of CUNY — We use molecular dynamics simulations to investigate the impact of a nanometer-sized drop onto structured atomic surfaces. Rough surfaces with Gaussian or power-law correlations are constructed using a Fourier synthesis algorithm. At low impact velocity drops spread into a lamella, and we study its shape and maximum extension as a function of surface roughness and wettability. At higher impact velocities a prompt splash occurs, and we examine the effects of the surface and external vapor on the behavior of the lamella rim. We also consider the effect of surface wettability patterns on splashing and spreading, and compare the results to lattice-Boltzmann simulations in the same geometry.

Sunday, November 18, 2012 4:45PM - 5:50PM –

Session E31 Focus Session: Interfacial Engineering in Thermal-Fluids II 33B - Chair: Kripa Varanasi, Massachusetts Institute of Technology

4:45PM E31.00001 Condensation enhancement using Liquid Impregnated Surfaces , SUSHANT ANAND, ADAM PAXSON, RAJEEV DHIMAN, J. DAVID SMITH, KRIPA VARANASI, Massachusetts Institute of Technology, VARANASI GROUP TEAM — Controlling the surface morphology of a low surface energy solid surface by imparting nano/micro-textures leads to water droplets existing in a Cassie-Baxter state characterized by minimal pinning of droplets, resulting in extremely low adhesion and low roll-off angles. These benefits are diminished during condensation as droplets form in the Wenzel state, causing high adhesion and extremely low shedding rates. Here we show that condensing on a hybrid surface comprised of a rough solid surface and an encapsulating liquid immiscible to water results in shedding of droplets much smaller than capillary length of water. It is shown that such hybrid surfaces have high nucleation capability usually associated with hydrophilic surfaces. The spreading coefficient of encapsulating liquid on water plays a crucial role during the condensation process. The surface morphology of the solid fraction of the hybrid surface needs to be adapted to stabilize encapsulating liquid on its surface and reduce contact between the solid surface and condensing droplets. This can be achieved by imparting nano/micro-textures on the surface.

4:58PM E31.00002 Novel microfluidic diodes: one-way wicking in open micro-channels controlled by channel topography[1] , JIANSHENG FENG, JONATHAN ROTHSTEIN, UMass Amherst, UMASS AMHERST TEAM — A series of open microfluidic channels with specially designed asymmetric internal structures were fabricated using standard photo-lithography and soft molding techniques. These micro-channels were shown to produce an asymmetric wicking behavior for a series of IPA-water mixtures. In some cases, the test liquids were found to completely wick in one direction while not wicking at all in the opposite direction. The wicking speed and degree of asymmetry was affected by the contact angle of the wicking fluid and the specific geometry of the angled fin-like structures added to the sides of the micro-channels. Surface-tension effects induced by the presence of the channels' internal structures were found to be the dominating physical mechanism responsible for the observed wicking behavior. Numerical simulations were performed to investigate the interface profile developed by the liquid front as it wicked through the channels. These simulations showed that three-dimensional effects were important in determining the extent and speed of wicking in these micro-channels. The findings of this study are expected to provide a better understanding of how fluids interact with micro-scaled structures and to offer a new way of manipulating fluids at the micron and nanometer scale.

[1]This work was supported by National Science Foundation's Center for Hierarchical Manufacturing (Grant number: CMMI-1025020)

5:11PM E31.00003 Leidenfrost Vapor Layer Stabilization on Superhydrophobic Surfaces , IVAN VAKARELSKI, King Abdullah University of Science and Technology, NEELESH PATANKAR, Northwestern University, JEREMY MARSTON, King Abdullah University of Science and Technology, DEREK CHAN, University of Melbourne, SIGURDUR THORODDSEN, King Abdullah University of Science and Technology — We have performed experiments to investigate the influence of the wettability of a superheated metallic sphere on the stability of a thin vapor layer during the cooling of a sphere immersed in water. For high enough sphere temperatures, a continuous vapor layer (Leidenfrost regime) is observed on the surface of non-superhydrophobic spheres, but below a critical sphere temperature the layer becomes unstable and explosively switches to nuclear boiling regime. In contrast, when the sphere surface is textured and superhydrophobic, the vapor layer is stable and gradually relaxes to the sphere surface until the complete cooling of the sphere, thus avoiding the nuclear boiling transition altogether. This finding could help in the development of heat exchange devices and of vapor layer based drag reducing technologies.

5:24PM E31.00004 Viscoelasticity measurement of gel formed at the liquid-liquid reactive interfaces , TOMOHIRO UJIIE, YUTAKA TADA, SHUICHI IWATA, YOSHIHITO KATO, Nagoya Institute of Technology, YUICHIRO NAGATSU, Tokyo University of Agriculture and Technology — We have experimentally studied a reacting liquid flow with gel formation by using viscous fingering (VF) as a flow field. Here, two systems were employed. In one system, sodium polyacrylate (SPA) solution and ferric ion solution were used as the more and less viscous liquids, respectively. In another system, xthantan gum (XG) solution and the ferric ion solution were used as the more and less viscous liquids, respectively. We showed that influence of gel formation on VF were qualitatively different in these two systems. We consider that the difference in the two systems will be caused by the difference in the properties of the gels. Therefore, we have measured the rheological properties of the gels by means of a rheometer. In the present study, viscoelasticity measurement was performed by two methods. One is the method which uses Double Wall Ring sensor (TA instrument) and another is the method using parallel plate. In both viscoelasticity measurements, the behavior of the formed gel was qualitatively consistent. We have found that the gel in the SPA system shows viscoelastic fluid like behavior. Moreover, we have found that the gel in the XG system shows solid like behavior.

5:37PM E31.00005 Seeing Below the Drop: Direct Nano-to-microscale Imaging of Complex Interfaces involving Solid, Liquid, and Gas Phases , KONRAD RYKACZEWSKI, Department of Mechanical Engineering, MIT & NIST, TREVAN LANDIN, FEI Co., MARLON L. WALKER, JOHN HENRY J. SCOTT, NIST, KRIPA K. VARANASI, Department of Mechanical Engineering, MIT — Nanostructured surfaces with special wetting properties have the potential to transform number of industries, including power generation, water desalination, gas and oil production, and microelectronics thermal management. Predicting the wetting properties of these surfaces requires detailed knowledge of the geometry and the composition of the contact volume linking the droplet to the underlying substrate. Surprisingly, a general nano-to-microscale method for direct imaging of such interfaces has previously not been developed. Here we introduce a three dimensional imaging method which resolves this one-hundred-year-old metrology gap in wetting research. Specifically, we demonstrate direct nano-to-microscale imaging of complex fluidic interfaces using cryofixation in combination with cryo-FIB/SEM. We show that application of this method yields previously unattainable quantitative information about the interfacial geometry of water condensed on silicon nanowire forests with hydrophilic and hydrophobic surface termination in the presence or absence of an intermediate water repelling oil. We also discuss imaging artifacts and the advantages of secondary and backscatter electron imaging, Energy Dispersive Spectrometry (EDS), and three dimensional FIB/SEM tomography.

Sunday, November 18, 2012 4:45PM - 5:50PM –

Session E32 Focus Session: Vortex Dynamics in Fluid-Structure Interactions III 33C - Chair: Silas Alben, University of Michigan

4:45PM E32.00001 Numerical Simulation of the Dynamic FSI Response and Stability of a Flapping Foil in a Dense Fluid , EUN JUNG CHAE, DENIZ TOLGA AKCABAY, YIN LU YOUNG, Department of Naval Architecture and Marine Engineering in University of Michigan — To advance the understanding of fish locomotion, improve the design biological devices or marine propulsions or turbines, or to explore innovative ocean energy harvesting ideas, it is important to be able accurately predict the dynamic fluid structure interaction (FSI) response and stability of flexible structures in a dense fluid. The objectives of this research are to (1) present an efficient and stable algorithm for numerical modeling of the dynamic FSI response and stability of a flapping foil in dense fluid, and (2) investigate the influence of fluid-to-solid density ratio on the FSI response and stability of a flapping foil. The numerical model involves coupling an unsteady RANS solver with a 2DOF structural model using a new hybrid coupling approach. The results show that the new hybrid coupling approach converge much faster than traditional loosely and tightly coupled approaches, and is able to avoid numerical instability issues due to virtual added mass effects for light, flexible structures in incompressible flow. The influence of density ratio on the FSI response, divergence and flutter speeds are presented, along with comparisons between viscous and inviscid FSI computations.

4:58PM E32.00002 A parametric study of thrust and efficiency of an oscillating airfoil , A.W. MACKOWSKI, C.H.K. WILLIAMSON, Cornell University — An oscillating airfoil serves as a classic test case for a variety of unsteady phenomena in fluid mechanics. In nature, fish, birds, and insects oscillate their fins and wings to produce thrust and maneuvering forces, often studied by approximating the appendages as airfoils. On the other hand, the unsteady fluid mechanics of an oscillating airfoil involve vortex shedding and vortex advection, which are essential to understanding unsteady thrust, and worth studying in their own right. This information is useful in areas such as flow control, fluid-structure interaction, and undersea robotics. In this work, we examine the thrust and efficiency of a heaving (or pitching) foil as a function of variables such as the reduced frequency and amplitude (noting previous related studies such as Koochesfahani 1989; Anderson et al. 1998). Further, our novel experimental "cyber-physical" technique [Mackowski & Williamson, 2011] allows the airfoil to propel itself under its own thrust. Our experimental apparatus constantly monitors the fluid forces acting on the foil, and commands velocity to a carriage system in accordance with these forces. With this capability, we are able to measure the terminal velocity of a self-propelled airfoil, as well as its stationary thrust and efficiency.

5:11PM E32.00003 Aerodynamic cause of the asymmetric wing deformation of insect wings[1] , HAOXIANG LUO, FANGBAO TIAN, JIALEI SONG, Vanderbilt University, XI-YUN LU, USTC — Insect wings typically exhibit significant asymmetric deformation patterns, where the magnitude of deflection during upstroke is greater than during downstroke. Such a feature is beneficial for the aerodynamics since it reduces the projected wing area during upstroke and leads to less negative lift. Previously, this asymmetry has been mainly attributed to the directional bending stiffness in the wing structure, e.g., one-way hinge, or a pre-existing camber in the wing surface. In the present study, we demonstrate that the asymmetric pattern can also be caused by the asymmetric force due to the flow, while the wing structure and kinematics are symmetric. A two-dimensional translating/pitching wing in a free stream is used as the model, and the wing is represented by an elastic sheet with large displacement. The result shows that, interestingly, the wing experiences larger deformation during upstroke even though the aerodynamic force is greater during downstroke. The physical mechanism of the phenomenon can be explained by the modulating effect of the aerodynamic force on the timing of storage/release of the elastic energy in the wing.

[1]Supported by NSF (No. CBET-0954381)

5:24PM E32.00004 Aerodynamic performance of membrane wings with adaptive compliance , OSCAR M. CURET, ALEXANDER CARRERE, ARJUN PANDE, KENNETH S. BREUER, Brown University — Some flying animals use wing membranes with adaptive compliance to control their aerodynamic performance. In this work we characterize the mechanical properties and aerodynamic performance of a low aspect ratio membrane wing composed of a dielectric film supported on a rigid frame. We test the wing model in a wind tunnel. When a fixed voltage is applied across the wing membrane the camber increases, accompanied by a small increase in lift (less than 2%). However, lift is significantly increased when the wing is forced with an oscillating field at specific frequencies that correspond to the characteristic vortex shedding frequency. We present the results concerning the kinematics and aerodynamic performance of the adaptive wing membrane and the coupling between the vortex shedding and the forced modulation of elastic modulus.

5:37PM E32.00005 Vortex interactions with membrane wings , RYE M. WALDMAN, KENNETH S. BREUER, Brown University — Membrane wings are common in flying animals such as bats, as well as in low Reynolds number Micro Air Vehicles. Vortices shed from the sharp leading- and trailing-edges and wing-tips of membrane wings, and the vortex interactions with the membrane play an important role in the wing's performance. When looking at compliant membrane wings that are initially tension-free at rest, there are two issues to consider: the static relationship between the net aerodynamic forces and the bulk wing deformation and the interaction between the membrane dynamics and unsteady flow structures. Nonlinear membrane deformation affects the membrane vibration modes, which in turn affects the coupling between the membrane and vortex shedding. We present coupled force, kinematic, and flow field measurements on low aspect ratio membrane wings of different thickness, with and without wing-tip support, over a range of angles of attack and freestream velocities. Wings with different tip support but of similar stiffness show similar static behavior, but different wing dynamics result in markedly different behavior in both the unsteady forces and the character of stall at high incidence angles.

5:50PM - 5:50PM –

Session F1 Poster Session (5:50PM-6:50PM) Ballroom 20D Foyer - Chair: Juan Carlos del Alamo, University of California, San Diego

F1.00001 MICROFLUIDICS –

F1.00002 Film thickness between a bubble and the inner wall of vertical tubes containing viscous liquids , GUADALUPE GUTIERREZ, ABEL LOPEZ-VILLA, ABRAHAM MEDINA, National Polytechnic Institute — We study numerically the film thickness in between the free surface of a bubble and the inner wall in a vertical tube filled with a viscous liquid. The computations were performed using the Boundary element method to solve the Stokes equations and a fourth order Runge-Kutta scheme to build the bubble shape. After the computation of the bubble shape, the thickness of the annular film was calculated for low Bond numbers and a wide range of Capillary numbers, Ca. For the case Ca close to zero (inviscid approximation) it is found that the film actually touches the wall, meanwhile for the viscous cases always there is a film of finite thickness.

F1.00003 Self-introduced thermocapillary convection in the evaporating process of a suspending drop , JINGCHANG XIE, Institute of Mechanics, CAS, NML TEAM — We studied the evaporation process of an evaporating ethanol drop coupled with self-introduced thermocapillary convection. The drop was suspended on the bottom of a rod to reduce the effect of buoyancy. We used different particle tracers to visualize the interior flow field and the gaseous exterior of the drop and measured the temperature distributions inside and outside the evaporating drop. Both good heat conductor and heat insulating material were used as the rod materials to investigate their effects on the surface temperature distribution of an evaporating drop. In the case of a copper rod with ethanol drop in ambient temperature, temperature gradient existed on the drop surface which results in a stable thermocapillary convection and cells appeared near the surface in the drop throughout entire evaporating process. The convection greatly changed the temperature distribution and the way energy and mass transfer. Temperature discontinuity was found existing at the surface of the evaporating drop. The boundary condition at the surface of the evaporating drop can lead to gaseous exterior convective transport. Because of the existence of thermocapillary convection, drop evaporating process or evaporating rate is enhanced.

F1.00004 Hole-Closing of a Surfactant Layer on a Thin Fluid Film , MATTHEW HIN, M. RICHARD SAYANAGI, RACHEL LEVY, Harvey Mudd College, KAREN DANIELS, NC State University — The spreading of surfactants on a thin fluid layer has been most commonly studied in an outward-spreading geometry. We report experiments on the inverse, the inward spreading of surfactant into a clean disk-shaped region, known as hole-closing. We observe that the inward force produces a transient distention, in which the underlying fluid is raised within the closing region. Using a laser line to image the height profile of the fluid surface, we characterize the height and evolution of the fluid distention. We observe that the height of the distension is controlled by a combination of fluid depth, surface tension difference, and chemical composition of both fluid and surfactant. Once formed, the height of the distension decays approximately exponentially, with the timescale primarily set by the particular choice of surfactant and underlying fluid.

F1.00005 Investigation of thin film coating process for printed electronics with suspension ink by slot die coating , SEUNG-HYUN LEE, INYOUNG KIM, Korea Institute of Machinery & Materials — Slot die coating process can be easily combined with roll-to-roll process and handle various coating liquid with wide range of viscosity and solid content. It is also pre-metered coating and the thickness of the coated layer can be easily predicted and controlled by a given feed flow rate and coating speed. Therefore, recently, slot die coating process is extending the use of fabrication of thin film printed electronics such as transparent conductive film and thin film solar cell etc. In the present study, we elucidated thin film coating process for printed electronics with suspension ink by slot die coating. Numerical study was investigated the effect of coating die design and rheological characteristics of suspension ink on coating uniformity. Slot die coating experiments was also performed with suspension ink which is composed of Cu(InGa)Se2 nano-particle and ethanol solvent and compared with numerical simulation.

F1.00006 Molecular dynamics simulations of the evaporation of particle-laden drops , WEIKANG CHEN, JOEL KOPLIK, ILONA KRETZSCHMAR, City College of CUNY — We use molecular dynamics simulations to study the evaporation of particle-laden droplets on a heated surface. The droplets are composed of a Lennard-Jones fluid containing rigid particles which are spherical sections of an atomic lattice, and heating is controlled through the temperature of an atomistic substrate. We observe that sufficiently large (but still nano-sized) particle-laden drops exhibit contact line pinning, measure the outward fluid flow field which advects particle to the drop rim, and find that the structure of the resulting aggregate varies with inter-particle interactions. In addition, the profile of the evaporative fluid flux is measured with and without particles present, and is also found to be in qualitative agreement with earlier theory. The compatibility of simple nanoscale calculations and micron-scale experiments indicates that molecular simulation may be used to predict aggregate structure in vaporative growth processes.

F1.00007 Non-Linear Oscillation in Ionic Current Due to Size Effect in Glass Nanopipette[1] , TOMOHIDE TAKAMI, XIAO LONG DENG, JONG WAN SON, EUN JI KANG, TOMOJI KAWAI, BAE HO PARK, Department of Physics, Konkuk University — We studied the size effect of the ionic current in glass pipette, and found an interesting 2.7 mHz oscillation at 50 nm. In this study, we would like to discuss the mechanism of the non-linear oscillation. Cation-rich layer with its Debye length λ exists in nanopipette, and its conductivity σ_d is lower than that in the central bulk layer σ_b in this study. The pressure difference $\Delta P = \Delta cRT$ where Δc is the difference in concentrations between in and out of the pipette. Then, the ionic current I can be estimated by using Hagen-Poiseuille equation;

$$I = \frac{\pi}{8\eta}\frac{\Delta cRT}{\ell}\left\{\sigma_d r^4 + (\sigma_b - \sigma_d)(\lambda - r)^2\left(r^2 + 2r\lambda - \lambda^2\right)\right\}.$$

(r: inner radius, ℓ: pipette length, η: viscosity) The last term indicates the non-linear oscillation. Moreover, we roughly estimated $\lambda = 2.08\times(2r)^{1/2}$. Then, the bulk layer appears appropriately when $2r \sim$50 nm, which causes the effective ionic current oscillation.

[1]This work was supported by KOSEF NRL Program grant funded by the Korea Government MEST (Grant No. 2010-0024525 and R0A-2008-000-20052-0), and WCU Program through the KOSEF funded by the MEST (Grant No. R31-2008-000-10057-0).

F1.00008 Resolving distinct conformations of spectrally similar silver-DNA nanoclusters using electrokinetic flows , JACKSON DEL BONIS-O'DONNELL, University of California Santa Barbara, DEBORAH FYGENSON, Department of Physics, University of California Santa Barbara, SUMITA PENNATHUR, Department of Mechanical Engineering, University of California Santa Barbara — Silver-DNA nanoclusters (Ag:DNA) are hybrid fluorescent macromolecules in which a silver superatom is stabilized by segments of single stranded DNA in aqueous solution. Recently, electrokinetic separations in microchannels have proven useful for measuring the size and charge of different Ag:DNA emitters stabilized by the same sequence of DNA. Small (50-100pL) fluorescent sample plugs are electrokinetically injected down a 30 mm long, 20 μm deep silica channel in the presence of a buffered background-electrolyte. Fluorophores contained within the injected plug travel at different velocities and thus separate down the length of the channel due to their differences in electrophoretic mobility. Diffusion measurements are also performed in situ by watching the time evolution of a stationary fluorescent sample plug. In the current work, the above techniques are applied to Ag:DNA stabilized by different sequences of DNA designed to adopt similar structures: a 12 cytosine single-stranded loop. Microfluidic separation measurements reveal the presence of multiple, spectrally similar Ag:DNA for different sequences, distinguished by their electrophoretic mobilities. Our results show that both versions of the 12C hairpin motif produce multiple fluorescent species each with different

F1.00009 Stochastic Analysis of Antibody-antigen Binding in a Microfluidic Device[1] , SHAUNA ADAMS, CONG ZHANG, HARVEY ZAMBRANO, A.T. CONLISK, The Ohio State University — Over the last decade, microfluidic "Labs on a Chip" (LOC) have evolved from a single microchannel to micro-total analysis systems (TAS) capable of integrating thousands of reaction vessels, conduits and valves-the contents of an entire chemical laboratory-on a single chip. These systems have several advantages in biomedical applications, including lower equipment and personnel costs, reduced power requirements, faster separations, and smaller sample and reagent volume requirements. Circulating tumor cells (CTC) are cancer cells found in the blood stream indicating the presence of a tumor in the body. We consider the population of magnetically tagged antibodies to be characterized by a collection of stochastic trajectories; the probability of finding an antibody at a given position is assumed to be defined by the Fokker-Planck equation. The first objective is to determine the probability that one or more magnetically labeled antibodies will assume a trajectory that is within the neighborhood of a given cancer cell. Once this occurs the binding process can be described using a deterministic analysis and the modeling of this process is the second objective of the paper.

[1]Supported by the NSF Nanoscale Science and Engineering center (NSEC) for the Affordable Nanoengineering of Polymeric Biomedical Devices EEC-0914790.

F1.00010 Understanding the nonlinear-optical response of a liquid-core photonic-crystal fiber[1] , ILYA FEDOTOV, Department of Neuroscience, Kurchatov Institute National Research Center, Moscow, Russia, VLADIMIR MITROKHIN, Center of Photochemistry, Russian Academy of Sciences, ALEXANDER VORONIN, ANDREY FEDOTOV, DMITRIY SIDOROV-BIRYUKOV, ALEKSEY ZHELTIKOV, Physics Department, International Laser Center, M.V. Lomonosov Moscow State University — Liquid-core waveguide structures have long been known and intensely used in nonlinear optics. Photonic-crystal fiber (PCF) technologies enhance performance and offer new functionalities of liquid filled waveguides as tools for nonlinear optics. We demonstrate a hollow core PCF that supports single-mode guiding at wavelengths longer than 600 nm in a 4-μm-diameter liquid-filled core, thus offering an attractive platform for nonlinear-optical experiments in the liquid phase. This PCF is employed to demonstrate that liquid-phase materials can radically modify the nonlinear-optical response of a waveguide structure relative to a typical nonlinear response of a silica waveguide. We show that the strong inertia of optical nonlinearity, characteristic of highly nonlinear liquid-phase materials, gives rise to a pulse-width dependent spectral red shift of the spectrally broadened fiber output. This wavelength shift remains strong even for pulse widths as large as several hundreds of femtoseconds.

[1]This work was supported by the Russian Foundation for Basic Research, project 11-04-12185-ofi-m

F1.00011 On-chip Micro- and Nanofluidic Electrokinetic Injection and Separation for PEGylation Analysis[1]

, ELIJAH SHELTON, MARY BAUM, DAN MORSE, SUMITA PENNATHUR, University of California, Santa Barbara, PENNATHUR NANOFLUIDICS LABORATORY COLLABORATION, MORSE LABORATORY COLLABORATION — We present an experimental study of micro- and nanofluidic electrokinetic injection and separation in borosilcate channels as a method for characterizing size and zeta potential of biomolecules–specifically polyethlylene glycol (PEG), keyhole limpet hemocyanine (KLH), and pegylated KLH. While pegylation (the conjugation of proteins with PEG) is an established technique for enhancing a protein's therapeutic properties, reliable characterization of these conjugations by traditional analysis techniques (i.e. gel-electrophoresis, zetasizer) remains a challenge. Using a three-step electrokinetic sequence (load, gate, and inject), FITC labeled species and a fluorescein tracer dye are injected into a channel where they separate according to differences in electrophoretic mobility. We find the average absolute mobility of pegylated subunit KLH in 1 micron channels to be 56% that of unpegylated subunit KLH. In a 250 nm channel, we measure a 33% shift in the average absolute mobility of PEG dendrimers as compared to measurements in a 1 micron channel. These results begin to demonstrate how a micro- and nanofluidic-based approach might address the demand for effective and accessible nanoparticle characterization platforms.

[1]Supported by the Institute for Collaborative Biotechnologies

F1.00012 The Clogging Behavior of Tapered Microchannels

, SORELL MASSENBURG, KAARE JENSEN, DAVID WEITZ, Harvard University — Nearly every application involving fluid relies upon filtration, yet filter design is not well understood. Design features, such as shape and pore size distribution, can be modeled in two dimensions using soft lithographic techniques to fabricate microscale pores in polydimethylsiloxane. We are then able to characterize the efficacy of variations in pore design by clogging these pores with polystyrene microparticles stabilized by carboxyl surface groups. Previous studies show that a probabilistic model based upon the Poiseuille Law well describes straight microfluidic channels clogged via hard spheres that are smaller than the channel. This model predicts that clogging behavior determined by the smallest dimension of the pore (the constriction). We show that tapered microfluidic channels modeled after pores in filter membranes do not follow this model and investigated the differences. Contrasting our results to the aforementioned probabilistic model helps elucidate the function of certain filter design features.

F1.00013 Fluid Surface Deformation by Objects in the Cheerios Effect

, KHOI NGUYEN, MICHAEL MILLER, SHREYAS MANDRE, Brown University, MANDRE LAB TEAM — Small objects floating on a fluid/air interface deform of the surface depending on material surface properties, density, and geometry. These objects attract each other through capillary interactions, a phenomenon dubbed the "cheerios effect." The attractive force and torque exerted on these objects by the interface can be estimated if the meniscus deformation is known. In addition, the floating objects can also rotate due to such an interaction. We present a series of experiments focused on visualizing the the motions of the floating objects and the deformation of the interface. The experiments involve thin laser-cut acrylic pieces attracting each other on water in a large glass petri dish and a camera set-up to capture the process. Furthermore, optical distortion of a grid pattern is used to visualize the water surface deformation near the edge of the objects. This study of the deformation of the water surface around a floating object, of the attractive/repulsive forces, and of post-contact rotational dynamics are potentially instrumental in the study of colloidal self-assembly.

F1.00014 Shape Control of Doctor blade coated Polymer Electrodes via Microflow Control in a Drying Droplet[1]

, YUNSEOK JANG, JEONGDAI JO, SEUNG-HYUN LEE, Korea Institute of Machinery & Materials, PEMS TEAM — We demonstrated a simple patterning method for polymer electrodes such as poly(3,4-ethylenedioxythiophene)/poly(styrenesulfonate) (PEDOT/PSS) by using the doctor blade coater. We controlled the surface tension for controlling the polymer electrodes shape. We made use of the difference in wettability between hydrophobic surfaces and hydrophilic surfaces to make the polymer electrodes patterns. However, the polymer electrodes patterns made with our patterning method created undesirable ring-like stains, which were caused by the outward flow of the solute within the PEDOT/PSS solution drop. To achieve homogenous device performance, we proposed a simple process for removing this ring-like stain by making the surface tension gradient in the PEDOT/PSS solution drop. Because this surface tension gradient causes the inward flow of the solute within the PEDOT/PSS solution drop, the ring-like stain is removed. Finally, we confirmed the potential of our patterning method for polymer electrodes such as the PEDOT/PSS by fabricating pentacene thin-film transistors (TFTs) and measuring the electrical properties of the pentacene TFTs.

[1]This study was supported by a grant (B551179-08-03-00/ B551179-10-01-00/ NK167D/ SC0830) from the cooperative R&D Program funded by the Korea Research Council Industrial Science and Technology, Republic of Korea.

F1.00015 Electrical Power Generation by Mechanically Modulating Electrical Double Layers

, DONGYUN LEE, JONG KYUN MOON, JAEKI JEONG, HYUK KYU PAK, Department of Physics, Pusan National University, Busan 609-735, Korea — Many objects in contact with a liquid acquire some electronic charges on their surfaces. These charges on the surface attract counter ions from the liquid phase. This complex system is called electrical double layer (EDL). Since its geometry and structure is similar to an electric capacitor, it is also called an electrical double layer capacitor (EDLC). In this work we studied two EDLCs formed in a liquid droplet bridge between two parallel solid conducting plates. We found that when the bridge height was mechanically modulated, each EDLC was continuously charged and discharged generating an AC electric current across the plates. The results of this experiment can be useful for constructing a micro-fluidic power generation.

F1.00016 Direct numerical simulation of current-induced convection near an ion-selective surface

, CLARA DRUZGALSKI, MATHIAS B. ANDERSEN, ALI MANI, Stanford University — Understanding fundamentals of electrokinetic transport and fluid flow phenomena near ion-selective surfaces provides insight to improve systems such as electrodialysis for water deionization. The work of Rubinstein and Zaltzman [e.g. Phys Rev E 62, 2238 (2000)] have clarified qualitative aspects of how development of current-induced space-charge layers near ion-selective surfaces can lead to the onset of electro-osmotic instabilities. We expand on this work through multidimensional numerical simulation of the full nonlinear Poisson-Nernst-Planck and Navier-Stokes equations with ideally selective membrane boundary conditions. Our numerical scheme is optimized by exploiting the periodicity in the system parallel to the ion-selective surface, using a spectral method in these coordinates. In the wall normal direction a finite difference approach accurately captures the strongly nonlinear nested boundary layer structure. Our numerical scheme fully resolves the concentration profiles throughout the system including the numerically stiff electric double layer and extended space charge layer. Our simulations enable prediction of the full continuous current versus voltage curves showing overlimiting current without resorting to any adjustable parameter.

F1.00017 Soret-driven convection in colloidal suspensions

, MAHMOUD DARASSI, LAYACHI HADJI, The University of Alabama — Convection in colloidal suspensions of solid particles is characterized by the interplay between thermophoresis, sedimentation and Brownian diffusion. Their coupled effects is represented by a dimensionless parameter β and experiments by Chang *et al.* (2008) have shown that for a given set of experimental parameters, β is a function of the particle radius r_p with the function $\beta(r_p)$ having the shape of an inverted parabola with two roots in the range 5 nm $\leq r_p \leq 125$ nm. We investigate both the linear and non-linear convection in a suspension of solid particles using a particulate medium in a Rayleigh-Bénard geometry set-up. The analysis focuses on the particle dominated convection regime for which the onset is steady and to disturbances having infinitely long wavelength. For $0 < \beta \ll 1$, which corresponds to particle size near the two roots of $\beta(r)$, we retrieve the instability threshold conditions for the binary mixture model. For $\beta = O\big(1\big)$, we show that, unlike the binary mixture model, the conditions for instability onset can be mapped to corresponding experimental parameters. A nonlinear evolution equation is derived and its predictions compared to those of a similar equation for the binary mixture case.

F1.00018 Super free fall in a semi-infinite cone filled with a low viscosity liquid , AYAX TORRES, GUADALUPE GUTIERREZ, ABEL LOPEZ-VILLA, ABRAHAM MEDINA, National Polytechnic Institute — The super free fall of a low-viscosity liquid column, filling partially a section of a vertical conical tube of non-finite length, it is analyzed. Through the use of an inviscid and one-dimensional model for the flow we describe the complex simultaneous motion of the two liquid interfaces.

F1.00019 The thinning of viscous liquid threads.[1] , J. RAFAEL CASTREJON-PITA, ALFONSO A. CASTREJON-PITA, IAN M. HUTCHINGS, University of Cambridge — The thinning neck of dripping droplets is studied experimentally for viscous Newtonian fluids. High speed imaging is used to measure the minimum neck diameter in terms of the time τ to breakup. Mixtures of water and glycerol with viscosities ranging from 20 to 363 mPa s are used to model the Newtonian behavior. The results show the transition from potential to inertial-viscous regimes occurs at the predicted values of $\sim Oh^2$. Before this transition the neck contraction rate follows the inviscid scaling law $\sim \tau^{2/3}$. After the transition, the neck thinning tends towards the linear viscous scaling law $\sim \tau$.

[1]Project supported by the EPSRC-UK (EP/G029458/1) and Cambridge-KACST.

F1.00020 Spreading and Fragmenting Characteristics of Impacting Droplet on Micro/ Nanostructured Water-Repellent Surfaces[1] , HYUNGMO KIM, CHAN LEE, Department of Mechanical Engineering, MOO HWAN KIM, Division of Advanced Nuclear Engineering, JOONWON KIM, Department of Mechanical Engineering — After impact on a solid surface a droplet spreads, but in different ways such as deposition, rebound, and fragmentation. Because fragmentation occurs when the kinetic energy beat the surface energy during impact, Weber number is the most important dimensionless number in the rebound/fragmentation criteria. This also can be dramatically changed by the micro/nano-scale surface structures. Different micro/nanostructured surfaces were fabricated using silicon wet etching, black silicon formation, or the combination of these methods. Then, spreading and fragmenting events were analyzed with supporting experimental results. On the surfaces, the microstructures form obstacles to drop spreading and retracting, the nanostructures give extreme water-repellency, and the hybrid micro/nanostructures facilitate droplet fragmentation. Especially, the Cassie-Baxter's fraction factor of the microstructures can change rebound/fragmentation criteria. From this work, we finally investigate how the micro/nanostructures can change spreading and fragmenting dynamics during droplet impact.

[1]This work was supported by the National Research Foundation of Korea(NRF) grant funded by the Korea government (MEST) (No. 2011-0030075).

F1.00021 Enhanced two photon fluorescence microfluidic sensor based on dual cladding photonic-crystal fiber[1] , LYUBOV AMITONOVA, ILYA FEDOTOV, Department of Neuroscience, Kurchatov Institute National Research Center, ANDREY FEDOTOV, ALEKSEI ZHELTIKOV, Physics Department, International Laser Center, M. V. Lomonosov Moscow State University — The architecture of photonic-crystal fibers (PCFs) suggests a variety of strategies for optical sensing. A combination of TPA approaches with capabilities of fiber-optic probes offers numerous advantages, suggesting a convenient format for beam delivery, facilitating manipulation of excitation radiation, and allowing this excitation to be applied locally and selectively. In this work, we show that a PCF with a special design can realize different protocols of optical sensing, simultaneously serving, whenever necessary, for the collection and on-line monitoring of liquid-phase samples. Specially designed PCF is shown to substantially increase the guided-wave luminescent response from molecules excited through two-photon absorption (TPA) by femtosecond near-infrared laser pulses. Biophotonic implications of this waveguide TPL-response enhancement include fiber-format solutions for online monitoring of drug delivery and drug activation, interrogation of neural activity, biosensing, endoscopy, and locally controlled singlet oxygen generation in photodynamic therapy.

[1]This work was supported by the Russian Foundation for Basic Research, project 11-04-12185-ofi-m.

F1.00022 Speed of a Taylor Swimmer in Newtonian and Viscoelastic Fluids , MOUMITA DASGUPTA, Clark University, BIN LIU, Brown University, HENRY FU, University of Nevada, Reno, MICHAEL BERHANU, MSC Paris Diderot University, KENNETH BREUER, THOMAS POWERS, Brown University, ARSHAD KUDROLLI, Clark University — We demonstrate that the speed of an idealized Taylor swimmer with a prescribed waveform in a viscoelastic fluid can be greater or lesser than in a Newtonian fluid depending on their rheological properties. The measurements are performed using a cylindrical sheet immersed in a fluid inside a cylindrical tank under torque free conditions with traveling waves imposed in the azimuthal direction. Swimming speeds in the Newtonian case are found to be consistent with calculations using the Stokes equation. A faster swimming speed is found in a viscoelastic fluid which has a constant viscosity with shear rate. By contrast, a slower swimming speed is found with more complex shear thinning viscoelastic fluids which have multiple relaxation time scales. These results are compared with calculations with Olroyd-B fluids which find a decreasing swimming speed with Deborah Number given by the product of fluid elastic relaxation time scale and the driving frequency.

F1.00023 Interaction between shape and lipid flow in bilayer membranes , PADMINI RANGAMANI, UC Berkeley, ASHUTOSH AGRAWAL, University of Houston, KRANTHI MANDADAPU, Sandia National Laboratories, GEORGE OSTER, DAVID STEIGMANN, UC Berkeley — Biological membranes have unique mechanical properties: they are fluid in-plane but elastic in bending. The most popular continuum mechanics model of the lipid bilayer is the Helfrich model. This model has provided insight into many membrane phenomena. However, it is an equilibrium model for an elastic membrane and does not capture any dynamic effects. The theory of intra-surface viscous flow on lipid bilayers is developed by combining the equations for flow on a curved surface with those that describe the elastic resistance of bilayers to flexure. The model is derived directly from balance laws and thus augments alternative formulations based on variational principles. Conditions holding along an edge of the membrane are emphasized and the coupling between flow and membrane shape is simulated numerically. Simulations of the model show that membrane shape changes involve the interaction between flow and surface deformation that are intrinsically non-linear effects. In response to an applied lateral pressure normal to the membrane, the shape of the membrane changes. Simultaneously, the surface flow field evolves over time to accommodate the deformation. Using this model, we can study the formation of membrane tubes and many biological phenomena.

F1.00024 Oscillation-induced drop transport using low-frequency ac electrowetting[1] , JIWOO HONG, SEUNG JUN LEE, KWAN HYOUNG KANG, Department of Mechanical Engineering, Pohang University of Science and Technology — When a small drop starts to move contact angle hysteresis (CAH) manifests as a pinning force and hinders the drop transport on a solid surface. For such a reason, many researchers have been studying how to reduce CAH. On the contrary, we demonstrate that small drops on an inclined plane can be mobilized by utilizing CAH and drop oscillations driven by ac electrowetting rather than reducing CAH. By using the peculiar dependence of sliding velocity on the size of a drop and the applied ac frequency, we can selectively slide drops of a specific size or merge two volumetrically different drops along an inclined plane. In addition, the direction of drop motion is determined by the initial asymmetry of contact angles on both edges of a drop. Accordingly, small drops can climb up an inclined plane using low-frequency ac electrowetting when the initial asymmetry of contact angles is reversed. We also obtain the threshold voltage for a climbing drop and the empirical relationship between the applied voltage and the climbing velocity.

[1]This work was supported by the National Research Foundation of Korea (NRF) grant No.R0A-2007-000-20098-0 funded by the Korea government (MEST) and No. 20090083510 through Multi-phenomena CFD Engineering Research Center.

F1.00025 A pipette dispenses a charged droplet[1] , DONGWHI CHOI, HORIM LEE, Mechanical engineering, POSTECH, DO JIN IM, IN SEOK KANG, Chemical engineering, POSTECH, KWAN HYOUNG KANG, Mechanical engineering, POSTECH — Micropipettes are widely used in many scientific and engineering fields. However, it is hardly known that a droplet dispensed from a plastic pipette tip has a considerable amount of charges (order of 10-10 C). Here we report that the charged droplet is dispensed from a commercial and disposable plastic pipette tip and this charge is originated from the natural electrification between a solution and the inner surface of the pipette tip. The charge amount is dependent on not only the physicochemical properties of a solution (e.g., pH and a concentration) but also dispensing environments (e.g., atmospheric humidity and type of commercial pipette tip). To investigate the effects of the charge on the droplet dispensing, we calculate the electrical force between the droplet and the pipette tip though numerical simulation. The micropipette users especially, who are dealing with discrete droplets in their experiments, should consider this charge effect in their dispensing of a droplet.

[1]This work was supported by the National Research Foundation of Korea (NRF) Grant No. R0A-2007-000-20098-0 funded by the Korea government (MEST) and No. 20090083510 through Multiphenomena CFD Engineering Research Center.

F1.00026 An Experimental Study of the Effect of Viscosity on Bouncing Soap Droplets onto a Horizontal Soap Film[1] , AMY-LEE GUNTER, HOI DICK NG, Concordia University, Montreal, Canada — This experimental study aims to investigate the phenomenon of a bouncing soap droplet on a horizontal soap film, and how this behavior is affected by variations in the glycerol content of the solution for both the droplet and film. Direct visualization of the bouncing dynamics using high-speed photography allows determination of droplet size and rebound height as the viscosity is varied. In addition, the upper and lower limits of the mixture composition at which the viscosity of the fluid prevents the droplet from bouncing are determined. A thorough examination of this fluid trampoline was recently conducted by Gilet and Bush, the focus of which was to compare the effect of vibration in the soap film [T. Gilet and J.W.M. Bush, J. Fluid Mech. 625: 167-203, 2009]. A small amount of attention was given to the effect of viscosity changes in the droplet and film, and this work aims to expand on those findings.

[1]This work is supported by the Natural Sciences and Engineering Research Council of Canada (NSERC).

F1.00027 Nonlinear dynamics in a microfluidic loop device: Chaos and Fractals , JEEVAN MADDALA, RAGHUNATHAN RENGASWAMY, Texas Tech University — Discrete decision making and resistive interactions between droplets in a microfluidic loop device induces fascinating nonlinear dynamics such as multi-stability and period doubling. Droplets entering the device at fixed time intervals can exit at different periods or chaotically. One of the periodic behaviors that is observed in a loop is the three-period behavior; this is consistent with the notion that three period behavior implies chaos. Switching between these different dynamical regimes is achieved by changing the inlet droplet feeding frequency. Chaotic behavior is observed between islands of periodic behavior. We show through simulations and experimental observations that the transitions between periods are indeed chaotic. Network model is used to study the dynamic behavior for different inlet feeding frequencies resulting in the development of a bifurcation map. The bifurcation map shows that the three period dynamics is preceded by chaos. A Lyapunov exponent is used to further validate these results. The exit droplet spacing shows several fascinating patterns when the model is simulated for a large number of droplets in the chaotic regime. One such chaotic regime produces a fractal that has a boundary of cardioid. The correlation dimension for a fractal pattern produced by this particular loop system is calculated to be 0.7.

F1.00028 Accumulation of BSA in Packed-bed Microfluidics[1] , SAMANTHA SUMMERS, The Pennsylvania State University, CHUNTIAN HU, RYAN HARTMAN, University of Alabama — Alzheimers and Parkinsons are two diseases that are associated with protein deposition in the brain, causing loss of either cognitive or muscle functioning. Protein deposition diseases are considered progressive diseases since the continual aggregation of protein causes the patient's symptoms to slowly worsen over time. There are currently no known means of treatment for protein deposition diseases. Our goal is to understand the potential for packed-bed microfluidics to study protein accumulation. Measurement of the resistance to flow through micro-scale packed-beds is critical to understanding the process of protein accumulation. Aggregation in bulk is fundamentally different from accumulation on surfaces. Our study attempts to distinguish between either mechanism. The results from our experiments involving protein injection through a microfluidic system will be presented and discussed.

[1]Funding received by NSF REU Grant 1062611.

F1.00029 Measurement of the extensional viscosity for Newtonian and and viscoelastic liquids using the selective withdrawal technique , E. TREJO-PEIMBERT, R. ZENIT, Universidad Nacional Autonoma de Mexico, J.J. FENG, University of British Columbia — Extensional viscosity is hard to measure. None of the existing devices offers a definite solution to determine this property for both viscous and viscoelastic liquids. This is the main motivation of this investigation: to find an alternative and reliable method to measure this property. We propose the use of a device inspired in the well-known selective withdrawal system. A viscous fluid is withdrawn near the interface from below with a tube. The suction generates the deformation of the free surface and the flow beneath is largely extensional. We conducted measurements of the extension rate using PIV and measured the surface deformation to infer the extensional stress. By knowing these two quantities the value of the extensional viscosity is calculated. We present the measurements obtained for both viscous Newtonian and several non Newtonian fluids. For Newtonian fluids, we do obtain measurements of the Trouton ratio close to 3. We observe a variety of interesting behaviors for non Newtonian liquids, which will be presented and discussed.

F1.00030 Optoelectrokinetic trapping of Gold Nanoparticles for SERS Applications , AVANISH MISHRA, RAVIRAJ THAKUR, School of Mechanical Engineering, Purdue University, West Lafayette, Indiana, 47906, STUART WILLIAMS, Department of Mechanical Engineering, University of Louisville, Louisville, Kentucky 40292, ALOKE KUMAR, Oak Ridge National Laboratory, Oak Ridge, TN 37831-6445, STEVE WERELEY, School of Mechanical Engineering, Purdue University, West Lafayette, Indiana, 47906 — Gold or Silver nanoparticle based Microfluidic Surface Enhanced Raman Spectroscopy (SERS) provides an excellent platform for sensitive chemical and biomolecular detection. Microfluidic SERS requires metal nanoparticles to be accumulated in a controlled manner. In this work, we present an active optoelectrokinetic method for patterning gold nanoparticles on a planar electrode surface. This technique consists of two indium tin oxide (ITO) electrodes across which an AC electric field is applied. Gold nanocolloidal solution is filled between the electrodes. Projection of an infrared (1064 nm) laser beam on one of the electrode surfaces causes a perturbation in electrical conductivity and permittivity of the fluid which in turn creates microscopic flow instability and an electrothermal vortex is generated. The electrothermal vortex traps gold nanoparticles and brings them closer to the electrode surface by hydrodynamic drag force where nanoparticles are trapped by particle-electrode interactions. This leads to an accumulation of gold nanoparticles at the site of illumination which can be used as a SERS hot spot.

F1.00031 Experimental Results of Electrothermal Vortex in Parallel ITO-glass With High Conductivity Medium and AC electric field and Laser , JUPYOUNG KIM, JAE-SUNG KWON, STEVE WERELEY, Purdue University — In many previous works, diverse experiments about Electrothermal Vortex were done because ac dc electrokinetic manipulation of particles became an important research area and Electrothermal Vortex is a major part of the electrokinetics. Especially in this work, however, Electrothermal Vortex was observed in parallel two ITO-glass with high conductivity medium (the medium was potassium chloride (KCl)). Because of high conductivity of potassium chloride and micro characteristic length scale of ITO-glass, effect of joule heating is significant. Then, this influences on Electrothermal Vortex. In addition, the previous works were done by complicated form of electrodes in order to make non-uniform electric fields. Making theses electrode shape is too fastidious to do readily, while our work is very simple so that it is proper to any type of applications. According to Peak-to-Peak voltage, Frequency, and Laser power, we measured shape and velocity of Electrothermal Vortex, Basically, Electrothermal Vortex was not related with AC frequency, but was proportional to Peak-to-Peak voltage and Laser power.

F1.00032 BIOFLUIDS –

F1.00033 Simultaneous analysis of wake structure and force measurements of unsteady hydrofoils

, TIMOTHY JETER, JR., MELISSA GREEN, Syracuse University — In the field of bio-inspired hydrodynamics, comparison of simultaneous wake structure analysis and force measurement on the body itself are relatively unexplored. Hydrofoil technology in an experimental setup seeks to bridge this gap, with water tunnel experiments that utilize a pitching and heaving low-aspect-ratio airfoil mounted vertically. The actuation is accomplished by employing a two-axis motion controller. Vortical wake structure will be analyzed using particle image velocimetry (PIV), and hydrodynamic forces will be measured using a six-component force balance installed within the actuation system. The simultaneous measurements will allow for correlation of wake structure to propulsive power and efficiency and evaluate their relationship with variations in Strouhal number and Reynolds number.

F1.00034 Characterization of Surface Acoustic Wave Nebulization: Atomization dynamics and resulting droplet size distribution

, ALICIA CLARK, ALBERTO ALISEDA, University of Washington Department of Mechanical Engineering, SCOTT HERON, YUE HUANG, DAVID GOODLETT, University of Washington Department of Medicinal Chemistry — High-speed imaging and Phase Doppler Particle Analyzer (PDPA) measurements are used to characterize the size and velocity distributions of micron-sized droplets produced by a surface acoustic wave (SAW) microelectronic nebulizer. The effects of drop composition, electric field amplitude and pulsation frequency, and initial drop volume have been experimentally studied. We observe that the droplets created in pure water are smaller, $\sim$2 μm, and the plume more concentrated near the nebulizer, with small second probability peak for large diameters, $\sim$100 μm. Pure methanol droplets have larger diameters, $\sim$ 5 μm, and lower volume concentration in the nebulizer plume, as corresponds to less efficient atomization process. The influence of fluid viscosity and surface tension will be discussed. Measurements of the velocity distribution show a strong dependency with excitation amplitude and duty factor.

F1.00035 Thermodynamic of cellulose solvation in novel solvent mixtures

, RITANKAR DAS, University of California, Berkeley — Biomass contains abundant amounts of cellulose as crystalline microfibrils. A limiting step to using cellulose as an alternative energy source, however, is the hydrolysis of the biomass and subsequent transformation into fuels. Cellulose is insoluble in most solvents including organic solvents and water, but it is soluble in some ionic liquids like BMIM-Cl. This project aims to find alternative solvents that are less expensive and are more environmentally benign than the ionic liquids. All-atom molecular dynamics simulations were performed on dissociated glucan chains separated by multiple (4-5) solvation shells, in the presence of several novel solvents and solvent mixtures. The solubility of the chains in each solvent was indicated by contacts calculations after the equilibration of the molecular dynamics. It was discovered that pyridine and imidazole acted as the best solvents because their aromatic electronic structure was able to effectively disrupt the inter-sheet interactions among the glucan chains in the axial direction, and because perturbation of the solvent interactions in the presence of glucan chains was minimal.

F1.00036 A Computational Model of Optimal Vein Graft Adaptation in an Arterial Environment

, ABHAY B. RAMACHANDRA, SETHURAMAN SANKARAN, Department of Mechanical and Aerospace Engineering, UCSD, JAY HUMPHREY, Department of Biomedical Engineering, Yale University, ALISON MARSDEN, Department of Mechanical and Aerospace Engineering, UCSD — In coronary artery disease, surgical revascularization using venous bypass grafts is performed to relieve symptoms and prolong life. Coronary bypass graft surgery is performed on approximately 500,000 people every year in the United States, with graft failure rates as high as 50% within 5 years. When a vein graft is implanted in the arterial system it adapts to the high flow rate and high pressure of the arterial environment by changing composition and geometry, and thus stiffness. Hemodynamic loads, resulting in altered wall shear and intramural stresses, are major factors impacting vein graft remodeling. Here, a constrained mixture theory of growth and remodeling for arteries is extended to model the evolution of a vein graft subjected to arterial flow and pressure conditions. A derivative-free optimization method is used to estimate the optimal set of constitutive parameters that best match passive biaxial mouse inferior vena cava data from experiments. Optimization is performed using surrogate management framework, a pattern search method with established convergence theory. The resulting parameter set is used to predict optimal vein adaptation in an arterial environment for two illustrative cases: a) Step change b) Gradual change in loading. Results are compared against vein graft data from the literature and a possible set of mechanisms for sub-optimal vein graft remodeling is suggested.

F1.00037 Deformation of Congenital Bicuspid Aortic Valves in Systole

, KAI SZETO, MAE UCSD, PETER PASTUSZKO, Rady Children's Hospital; Department of Surgery UCSD, VISHAL NIGAM, Rady Children's Hospital; Department of Pediatrics UCSD, JUAN LASHERAS, Departments of MAE and Bioengineering, UCSD — Clinical studies have shown that patients with congenital bicuspid aortic valves (CBAVs) develop degenerative calcification of the leaflets at young ages compared to normal tricuspid aortic valves (TAVs). It has been hypothesized that the asymmetrical geometry of the leaflets in CBAVs and the associate changes in flow shear stresses and excessive strain rate levels are possible causes for the early calcification. Central to the validation of this hypothesis is the need to quantify the differences in strain rate levels between the BAVs and TAVs. We simulate the CBAVs by surgically stitching two of the leaflets of a porcine aortic valve together. To quantify strain differences, we performed in-vitro experiments in both BAVs and TAVs by tracking the 3-D motion of small dots marked on each leaflet surface. We then used phase-locked stereo photogrammetry to measure the strain rates in both radial and circumferential directions during the whole cardiac cycle. In the BAVs' case, the fused leaflet experiences an almost 30% increase in the radial stretching when fully open. RNA profiling of human aortic valve interstitial cells exposed to cyclic stretch shows that the increased stretch experienced by the BAVs results in increased levels of INTERLEUKINS (ILs) and other known inflammatory markers associated with aortic valve calcification. Together, these observations suggest that the abnormal stretch experienced by BAVs activates inflammation gene expression.

F1.00038 ABSTRACT WITHDRAWN –

F1.00039 Characterization of intraventricular flow patterns in healthy neonates from conventional color-Doppler ultrasound[1]

, SHAI TEJMAN-YARDEN, CALLIE RZASA, Dept of Cardiology, Rady Children's Hospital, San Diego, CA, YOLANDA BENITO, Dept of Cardiology, Gregorio Maranon Hospital, Madrid, Spain, MARTA ALHAMA, Dept of Internal Medicine, Scripps Green Hospital, La Jolla, CA, TINA LEONE, Dept. of Pediatrics, UC San Diego, RAQUEL YOTTI, JAVIER BERMEJO, Dept of Cardiology, Gregorio Maranon Hospital, Madrid, Spain, BETH PRINTZ, Dept of Cardiology, Rady Children's Hospital, San Diego, CA, JUAN C. DEL ALAMO, MAE Dept, UC San Diego, La Jolla, CA — Left ventricular vortices have been difficult to visualize in the clinical setting due to the lack of quantitative non-invasive modalities, and this limitation is especially important in pediatrics. We have developed and validated a new technique to reconstruct two-dimensional time-resolved velocity fields in the LV from conventional transthoracic color-Doppler images. This non-invasive modality was used to image LV flow in 10 healthy full-term neonates, ages 24-48 hours. Our results show that, in neonates, a diastolic vortex developed during LV filling, was maintained during isovolumic contraction, and decayed during the ejection period. The vortex was created near the base of the ventricle, moved toward the apex, and then back toward the base and LVOT during ejection. In conclusion, we have characterized for the first time the properties of the LV filling vortex in normal neonates, demonstrating that this vortex channels blood from the inflow to the outflow tract of the LV. Together with existing data from adults, our results confirm that the LV vortex is conserved through adulthood.

[1]Funded by NIH Grant R21HL108268.

F1.00040 Forewing-hindwing phase-lag effect in the propulsive performance of a four-winged flapping flyer[1] , MARIANA CENTENO[2], PMMH UMR7636 CNRS, ESPCI ParisTech, UPMC (Paris 6), U. Paris Diderot (Paris 7) and Facultad de Ciencias, UNAM, DANIEL PRADAL, BENJAMIN THIRIA, RAMIRO GODOY-DIANA, PMMH UMR7636 CNRS, ESPCI ParisTech, UPMC (Paris 6), U. Paris Diderot (Paris 7) — We study experimentally a four-winged flapping flyer with chord-wise flexible wings in a self-propelled configuration. For a given physical configuration of the flyer (i.e. fixed distance between the forewing and hindwing pairs and fixed wing flexibility), we explore the kinematic parameter space constituted by the flapping frequency and the forewing-hindwing phase lag. Net thrust force, cruising speed and consumed electric power measurements were performed for each point in the (f, φ) parameter space. These results are analyzed in parallel with two-dimensional velocity field measurements obtained by time-resolved particle image velocimetry around a forewing-hindwing pair.

[1]We acknowledge support from the French National Research Agency through project No. ANR-08-BLAN-0099 and of EADS Foundation through project "Fluids and elasticity in biomimetic propulsion".
[2]MC acknowledges support from CONACyT and UNAM, Mexico.

F1.00041 Three-dimensional flow around a hovering hummingbird[1] , JIALEI SONG, HAOXIANG LUO, Vanderbilt University, TYSON HEDRICK, University of North Carolina at Chapel Hill — We use an immersed-boundary method to simulate the complex three-dimensional flow around a hovering hummingbird and study the unsteady vortical structures in the flow. In the simulation, the realistic wing kinematics is reconstructed from high-speed imaging data of a Rufous hummingbird, and thus the wing surface does not assume a two-dimensional plane. The Reynolds number is approximately 3000 based on the average wing-tip velocity and the mean cord length. More than 16 million Cartesian mesh points are used in the simulation, which allows us to capture both near- and far-field vortices. We will show the detailed flow structures in the presentation and will compare the numerical result with previous experimental measurement. In addition, we will discuss the force characteristics and the aerodynamic power of the bird.

[1]Supported by NSF (No. CBET-0954381)

F1.00042 Understanding cell passage through constricted microfluidic channels[1] , MARCO A. CARTAS-AYALA, ROHIT KARNIK, Massachusetts Institute of Technology — Recently, several microfluidic platforms have been proposed to characterize cells based on their behaviour during cell passage through constricted channels. Variables like transit time have been analyzed in disease states like sickle cell anemia, malaria and sepsis. Nevertheless, it is hard to make direct comparisons between different platforms and cell types. We present experimental results of the relationship between solid deformable particle properties, i.e. stiffness and relative particle size, and flow properties, i.e. particle's velocity. We measured the hydrodynamic variables during the flow of HL-60 cells, a white myeloid cell type, in narrow microfluidic square channels using a microfluidic differential manometer. We measured the flow force required to move cells of different sizes through microchannels and quantified friction forces opposing cell passage. We determined the non-dimensional parameters that influence the flow of cells and we used them to obtain a non dimensional expression that can be used to predict the forces needed to drive cells through microchannels. We found that the friction force needed to flow HL-60 through a microfluidic channel is the sum of two parts. The first part is a static friction force that is proportional to the force needed to keep the force compressed. The second part is a factor that is proportional to the cell velocity, hence a dynamic term, and slightly sensitive to the compressive force.

[1]We thank CONACYT (Mexican Science and Technology Council) for supporting this project, grant 205899.

F1.00043 Continuous-flow separation of live and dead yeasts using reservoir-based dielectrophoresis (rDEP)[1] , SAURIN PATEL, DANIEL SHOWERS, PALLAVI VEDANTAM, TZUEN-RONG TZENG, Clemson University, SHIZHI QIAN, Old Dominion University, XIANGCHUN XUAN, Clemson University — Separating live and dead cells is critical to the diagnosis of early stage diseases and to the efficacy test of drug screening etc. We develop a novel microfluidic approach to continuous separation of yeast cells by viability inside a reservoir. It exploits the cell dielectrophoresis that is induced by the inherent electric field gradient at the reservoir-microchannel junction to selectively trap dead yeasts and continuously sort them from live ones. We term this approach reservoir-based dielectrophoresis (rDEP). The transporting, focusing, and trapping of live and dead yeast cells at the reservoir-microchannel junction are studied separately by varying the DC-biased AC electric fields. These phenomena can all be reasonably predicted by a 2D numerical model. We find that the AC to DC field ratio for live yeast trapping is higher than that for dead cells because the former experiences a weaker rDEP while having a larger electrokinetic mobility. It is this difference in the AC to DC field ratio that enables the viability-based yeast cell separation. The rDEP approach has unique advantages over existing DEP-based techniques such as the occupation of zero channel space and the elimination of in-channel mechanical or electrical parts.

[1]NSF

F1.00044 ABSTRACT WITHDRAWN –

F1.00045 Establishing live cell directional 2-point particle tracking microrheology , RICARDO SERRANO, MANUEL GOMEZ-GONZALEZ, JUAN C. DEL ALAMO, University of California, San Diego — Directionality is essential for cell function and relation with its environment: an isotropic cell would not be able to move, mechanotransduct or perform any action other than isotropic compression and expansion. Cell directionality is achieved through chemical gradients and mechanical orientation of the cytoskeleton. The directional mechanics of the cell cytoplasm is described by the Leslie-Ericksen equations, dependent on the 3 Miesowicz viscoelasticity coefficients. They can be measured by using Directional Particle Tracking Microrheology. However, the 3D motion of a particle in a nematic environment provides only 2 independent equations, and only 2 viscoelasticity coefficients could previously be calculated, i.e. by tracking a single particle, we could only fully describe a pseudo-isotropic fluid. In this study, we analyze the motion of two microspheres in a directional nematic fluid. The medium is composed of an elastic directional network viscously coupled to a viscous isotropic liquid. The correlated motion of the two microspheres provide 3 independent equations. These equations can be used to measure the 3 directional viscosity coefficients. We show that, by using Directional 2-Point PTM, we can determine the complete microrheology of a nematic material.

F1.00046 Comprehensive Spatial Display of the Microcirculation in a Capillary Bundle from Rat Spinotrapezius Muscle Fascia Tissue , NIKI YAMAMURA, FRANK JACOBITZ, University of San Diego, GEERT SCHMID-SCHÖNBEIN, University of California, San Diego — Previous investigations of the microcirculation in skeletal muscle have utilized a statistical display of fundamental hemodynamic variables without regard of actual microvascular details. The focus of this project is the development of a software tool to perform a spatial analysis of hemodynamic results. The vessel network considered in this study is a capillary bundle in rat spinotrapezius muscle fascia with transverse arterioles supplying blood, capillary vessels, and collecting venules removing blood. The software tool represents information about blood vessel location and connectivity in two matrices. The first matrix holds spatial locations of vessel intersections or nodes. This matrix is used to create a second matrix containing the locations of all microvessels. The second matrix is then used to produce result matrices holding the values of flow properties at the locations at which they are observed in the vessel network. The resulting images provide a full display, for example, of the pressure drop in the network. The highest velocities are obtained in the transverse arterioles and adjacent capillaries, while other vessels in the network show lower velocities. An area of elevated hematocrit is observed in the periphery of the network.

F1.00047 Simulations of the Microcirculation in the Human Conjunctiva , WILLIAM DOW, FRANK JACOBITZ, University of San Diego, PETER CHEN, University of California, San Diego — The microcirculation in the conjunctiva of a healthy human subject is analyzed using a simulation approach. A comparison between healthy and diseased states may lead to early diagnosis for a variety of vascular related disorders. Previous work suggests that hypertension, arteriosclerosis, and diabetes mellitus have noticeable very early changes in the microvasculature (Davis and Landau, 1957; Ditzel, 1968; Kunitomo, 1974) and the vessels of the conjunctiva are specifically useful for this research because they can be studied non-invasively. The microcirculation in the conjunctiva has been documented over the course of disease treatments, providing both still images and video footage for information on vessel length, diameter, and connectivity as well as the direction of blood flow. The numerical method is based on a Hagen-Poiseuille balance in the microvessels and a sparse matrix solver is used to obtain the solution. The simulations use realistic vessel topology for the microvasculature, reconstructed from microscope images of tissue samples, and consider blood rheology as well as passive and active vessel properties.

F1.00048 Proteins at flowing interfaces: From understanding structure to treating disease , DAVID POSADA, JAMES YOUNG, AMIR HIRSA, Rensselaer Polytechnic Institute — The field of soft matter offers vast opportunities for scientific and technological developments, with many challenges that need to be addressed by various disciplines. Fluid dynamics has a tremendous potential for greater impact, from broadening fundamental understanding to treating disease. Here we demonstrate the use of fluid dynamics in two biotechnology problems involving proteins at the air/water interface: a) 2-Dimensional protein crystallization and b) amyloid fibril formation. Protein crystallization is usually the most challenging step in X-ray diffraction analysis of protein structure. Recently it was demonstrated that flow can induce 2-D protein crystallization at conditions under which quiescent systems do not form crystals. A different form of protein structuring, namely amyloid fibrillization, is also of interest due to its association with several neurodegenerative diseases such as Alzheimer's and Parkinson's disease. Protein denaturation, which is the root of the fibrillization process, is also a significant concern in biotherapeutics production. Both problems are studied by using shearing free-surface flows in simple geometries. The common finding is that flow can significantly enhance the growth of protein structures.

F1.00049 Separation control effects of mako shark skin applied to NACA 4412 hydrofoil and a cylinder[1] , MICHAEL BRADSHAW, AMY LANG, University of Alabama, PHILIP MOTTA, MARIA HABEGGER, University of South Florida, ROBERT HUETER, Mote Marine Laboratory — Shark skin is investigated as a means of passive flow separation control due to its preferential flow direction and the potential for its scales to obstruct low-momentum backflow resulting from an adverse pressure gradient. In this study, the effect of the scales on flow reversal is primarily observed in a tripped turbulent boundary layer by comparing the flow over a NACA 4412 hydrofoil with a smooth, painted surface to that over the same hydrofoil with samples of mako shark skin affixed to its upper surface. Also, the effect of the scales on unsteady flow reversal is observed in laminar flow conditions for flow over a cylinder covered with mako shark skin. These samples were taken from the shark's flank region because the scales at that location have been shown to have the greatest angle of bristling, and thus the best potential for separation control. All sets of flow data in this study were obtained using Time-Resolved Digital Particle Image Velocimetry. The flow was primarily analyzed by means of the backflow coefficient (a value based on the percentage of time that flow in a region over the hydrofoil is reversed), average backflow magnitude, and the time history of instantaneous flow velocity values at specific points in the boundary layer over the hydrofoil models.

[1]Research funded by NSF REU Grant # 1062611.

F1.00050 DPPC: Is it ever Newtonian? , AMIR SADOUGHI, AMIR HIRSA, Rensselaer Polytechnic Institute, JUAN LOPEZ, Arizona State University — Understanding the intrinsic properties of lung surfactant constituent components is important for a scientific foundation and predictive models. The three major interfacial properties affecting the hydrodynamics are surface tension, surface shear viscosity μ^s, and surface dilatational viscosity κ^s. At small scales such as in the alveoli, the effects of the interfacial viscosities are comparable to those of surface tension gradients. The surface tension can be determined reliably using a variety of methods. There are also several techniques that lead to consistent measurements of surface shear viscosity. Recently, for DPPC (the most prevalent constituent of lung surfactant), Newtonian behavior was demonstrated for several monolayer phases, and non-Newtonian flow documented for other phases. On the other hand, the surface dilatational viscosity is difficult to quantify, and no systematic measurements have been reported. We utilize a free-surface cavity with fixed sidewalls and oscillatory driven floor to measure the response of the monolayer. The sum $\mu^s + \kappa^s$ is deduced from comparisons between the experiments and Navier-Stokes computations with Boussinesq-Scriven surface model. Subsequently, the independent measurements of μ^s are used to isolate κ^s.

F1.00051 Non-Dimensional Formulation of Ventricular Work-Load Severity Under Concomitant Heart Valve Disease[1] , MELODY DONG, University of California, San Diego, RACHAEL SIMON-WALKER, LAKSHMI DASI, Colorado State University — Current guidelines on assessing the severity of heart valve disease rely on dimensional disease specific measures and are thus unable to capture severity under a concomitant heart valve disease scenario. Experiments were conducted to measure ventricular work-load in an in-house in-vitro left heart simulator. In-house tri-leaflet heart valves were built and parameterized to model concomitant heart valve disease. Measured ventricular power varied non-linearly with cardiac output and mean aortic pressure. Significant data collapse could be achieved by the non-dimensionalization of ventricular power with cardiac output, fluid density, and a length scale. The dimensionless power, Circulation Energy Dissipation Index (CEDI), indicates that concomitant conditions require a significant increase in the amount of work needed to sustain cardiac function. It predicts severity without the need to quantify individual disease severities. This indicates the need for new fluid-dynamics similitude based clinical guidelines to assist patients with multiple heart valve diseases.

[1]Funded by the American Heart Association

F1.00052 Aortic emboli show surprising size dependent predilection for cerebral arteries: Results from computational fluid dynamics[1] , IAN CARR, Illinois Institute of Technology, ROBERT SCHWARTZ, Minneapolis Heart Institute Foundation, SHAWN SHADDEN, Illinois Institute of Technology — Cardiac emboli can have devastating consequences if they enter the cerebral circulation, and are the most common cause of embolic stroke. Little is known about relationships of embolic origin/density/size to cerebral events; as these relationships are difficult to observe. To better understand stoke risk from cardiac and aortic emboli, we developed a computational model to track emboli from the heart to the brain. Patient-specific models of the human aorta and arteries to the brain were derived from CT angiography from 10 MHIF patients. Blood flow was modeled by the Navier-Stokes equations using pulsatile inflow at the aortic valve, and physiologic Windkessel models at the outlets. Particulate was injected at the aortic valve and tracked using modified Maxey-Riley equations with a wall collision model. Results demonstrate aortic emboli that entered the cerebral circulation through the carotid or vertebral arteries were localized to specific locations of the proximal aorta. The percentage of released particles embolic to the brain markedly increased with particle size from 0 to ≈1–1.5 mm in all patients. Larger particulate became less likely to traverse the cerebral vessels. These findings are consistent with sparse literature based on transesophageal echo measurements.

[1]This work was supported in part by the National Science Foundation, award number 1157041.

F1.00053 Capture of Magnetic Nanoparticles in Simulated Blood Vessels: Effects of Proteins and Coating with Poly(ethylene glycol)[1] , JAIMEE ROBERTSON, Syracuse University, CHRISTOPHER BRAZEL, University of Alabama — Magnetic nanoparticles (MNPs) have applications in cancer treatment as they can be captured and localized to a diseased site by use of an external magnetic field. After localization, cancer treatments such as magnetically targeted chemotherapy and localized hyperthermia can be applied. Previously, our lab has shown that the percent capture of MNPs is significantly reduced when MNPs are dispersed in protein solutions compared to water or aqueous polymer solutions. The purpose of this study was to determine the effects of proteins on capture efficiency and to investigate the ability of poly(ethylene glycol), PEG, coatings to reduce aggregation of MNPs with proteins, allowing for a greater capture of MNPs in flow. Using Tygon® tubing to simulate blood vessels, a maghemite nanoparticle solution was pumped through a capture zone, where a magnetic field was applied. After passing through the capture zone, the fluid flowed to a spectrophotometer, which measured the absorbance of the solution. The introduction of proteins into the nanoparticle solution reduced the percent capture of MNPs. However, coating the MNPs with PEG aided in preventing aggregation and led to higher capture efficiencies in protein solutions. Additionally, the effects of capture length and protein exposure time were examined. It was found that a higher percent capture is attainable with a longer capture length. Furthermore, on a scale of hours, the percent capture is not affected by the protein exposure time.

[1]Funded by NSF REU Grant 1062611 and NIH NCI R21CA 141388.

F1.00054 Effect of Varying the Angle of Attack of the Scales on a Biomimetic Shark Skin Model on Embedded Vortex Formation , JENNIFER WHEELUS, AMY LANG, MICHAEL BRADSHAW, EMILY JONES, FARHANA AFROZ, The University of Alabama, PHILIP MOTTA, MARIA HABEGGER, University of South Florida — The skin of fast-swimming sharks is proposed to have mechanisms to reduce drag and delay flow separation. The skin of fast-swimming and agile sharks is covered with small teeth-like denticles on the order of 0.2 mm. 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. 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 has been shown that for an angle of attack of 90 degrees, vortices form within these cavities and impose a partial slip condition at the surface of the cavity. This experiment focuses on smaller angles of attack for denticle bristling, closer to the range thought to be achieved on real shark skin. A 3-D bristled shark skin model with varying angle of attack, embedded below a boundary layer, was used to study the formation of cavity vortices through fluorescent dye visualization and Digital Particle Image Velocimetry (DPIV). The effect of varying angle of attack on vortex formation will be discussed.

F1.00055 Swimming micro-robot powered by stimuli-sensitive gel , HASSAN MASOUD, ALEXANDER ALEXEEV, George W. Woodruff School of Mechanical Engineering, Georgia Institute of Technology — Using three-dimensional computer simulations, we design a simple maneuverable micro-swimmer that can self-propel and navigate in highly viscous (low Reynolds-number) environments. Our simple swimmer consists of a cubic gel body which periodically changes volume in response to external stimuli, two rigid rectangular flaps attached to the opposite sides of the gel body, and a flexible steering flap at the front end of the swimmer. The stimuli-sensitive body undergoes periodic expansions (swelling) and contractions (deswelling) leading to a time-irreversible beating motion of the propulsive flaps that propel the micro-swimmer. Thus, the responsive gel body acts as an "engine" actuating the motion of the swimmer. We examine how the swimming speed depends on the gel and flap properties. We also probe how the swimmer trajectory can be changed using a responsive steering flap whose curvature is controlled by an external stimulus. We show that the turning occurs due to steering flap bending and periodic beating. Furthermore, our simulations reveal that the turning direction can be regulated by changing the intensity of external stimulus.

F1.00056 Analysis of High Speed Liquid Jets Emitted from Needle Free Jet Injectors[1] , ROCCO PORTARO, AMY-LEE GUNTER, HOI DICK NG, Concordia University, Montreal, Canada — The replacement of the traditional hypodermic needle by needle free liquid jet injectors has been of great interest to the scientific community over recent years. This study utilizes a specially designed needle free injector in order to describe the behavior of high speed liquid jets. High speed photography is used to depict the injection process, as the jet emitted from the injector penetrates biological tissue. The penetration depth of the jet will be studied by varying parameter such as the jet diameter, geometry and power. This analysis will then be used in improving the performance of liquid free injectors by maintaining more consistent injection depths and minimizing the power required to penetrate human tissue. This in turn leads to painless injections with less risk of contamination and aids in making needle free liquid jet injectors a viable alternative to hypodermic needles.

[1]This work is supported by Fonds de recherche du Quebec - Nature et technologies.

F1.00057 An Experimental Study of Flow Separation over a Flat Plate with Transverse Grooves[1] , EMILY JONES, AMY LANG, The University of Alabama — A shark's scales help to reduce drag over its body by controlling boundary layer separation over its skin. It is theorized that the scales bristle when encountering a reversing flow, thereby trapping vortices between the scales, creating a partial slip condition over the surface and inducing turbulence augmentation in the boundary layer. In an attempt to replicate and study these effects, a spinning cylinder was used in a water tunnel to induce separation over a flat plate with 2 mm, square 2-D transverse grooves and sinusoidal grooves of the same size. The results were compared to tripped, turbulent boundary layer separation occurring over a flat plate without grooves using DPIV. The strength of the adverse pressure gradient was varied, and the observed delay in flow separation and other effects upon the boundary layer are discussed.

[1]Funding received by NSF REU grant 1062611.

F1.00058 Formation of clogs under ultrasound excitation: a microfluidic study , ERIN BARNEY, EMILIE DRESSAIRE, Trinity College — Acoustic waves and more specifically ultrasound are commonly used in microfluidic devices to focus, separate and mix particles. We study the influence of ultrasound on the formation of clogs of colloidal particles in microchannels. In particular we focus on the role played by the flow properties and the characteristics of the acoustic wave (frequency and amplitude). We show that the ultrasound excitation delays the formation of clogs and interpret our results with a simple force balance on the colloidal particle.

F1.00059 Characterization of Intracellular Streaming and Traction Forces in Migrating Physarum Plasmodia , SHUN ZHANG, JUAN C. DEL ALAMO, MAE Dept, UC San Diego, ROBERT D. GUY, Math Dept. UC Davis, JUAN C. LASHERAS, MAE Dept, UC San Diego — Physarum plasmodium is a model organism for cell migration that exhibits fast intracellular streaming. Motile amoeboid physarum plasmodia were obtained from dish cultures of Physarum Polycephalum, a slime mold that inhabits shady cool moist areas in the wild, such as decaying vegetable material. The migrating amoebae were obtained by cutting successively smaller pieces from the growing tips of the cultured parent mold, and their size ranged 0.2 to 0.5 mm. Single amoebae were seeded and let adhere on flexible polyacrilamide gels that were functionalized with collagen, contained 0.2-micron fluorescent beads, and were embedded in an aqueous medium. Soon after adhering to the gel, the amoeabe began crawling at about 1mm/hr. Joint time-lapse sequences of intracellular streaming and gel deformation were acquired respectively in the bright and fluorescent fields of a confocal microscope at 20X magnification. These images were analyzed using particle-tracking algorithms, and the traction stresses applied by the amoebae on the surface were computed by solving the elastostatic equation for the gel using the measured bead displacements as boundary conditions. These measurements provide, for the first time, a joint characterization of intracellular mass transport and the forces driving this transport in motile amoeboid cells.

F1.00060 GENERAL —

F1.00061 Vorticity Flux of Low-Speed Flow Over a Deforming Arc Airfoil , MAJID MOLKI, NEGIN SATTARI, Southern Illinois University Edwardsville — Vorticity flux generated by a deforming arc airfoil is investigated. The model is based on the finite-volume method for laminar incompressible flow of air over a circular-arc airfoil. A deforming-mesh approach is employed to accommodate the motion and deformation of the airfoil. The vorticity flux is evaluated at the surface of the airfoil for both non-deforming (rigid) and deforming airfoils. Complex flow features such as boundary layer flows, vortical structures, rolling vortices, and vortex layers are all present and have some degree of influence on the aerodynamic characteristics of the airfoil. It is shown that the vorticity flux at the surface is influenced by tangential pressure gradient and tangential acceleration of the airfoil. Since the pressure gradient and acceleration are non-uniform over the airfoil, their contributions strongly depend on position, and they result in a diffused distribution of vorticity flux magnitude over the surface. Vorticity flux into the fluid results in the formation and growth of vortices which are swept with the flow downstream.

F1.00062 Unsteady flow around impacting square cylinder , CHANGYOUNG CHOI, MAN YEONG HA, HYUN SIK YOON, Pusan National University — The problem on the flow resulting from the collision without rebound of square cylinder with a wall at Re=200 is investigated computationally using the DF/FD method with a finite volume method. Emphasis is on the case of a square cylinder impact by three-dimensional numerical simulation, but comparisons with the flow generated by the impact of a circular cylinder are included. A cylindrical body impacting on the wall produces two primary vortex rings. The primary vortex rings spread outward away from the body along the wall. This continues until stalling while lifting induced wall vortices into the primary vortices. For normal square cylinder impact, secondary vortices exhibit a three-dimensional instability. Comparison with the circular cylinder impact reveals that this is caused by the differences in flow strength after the cylinder collides with the wall. Oblique square cylinder impacts are also considered. For the oblique square cylinder, a three-dimensional instability does not appear in the flow around the cylinder. As the impact angle increases, the wall effect is gradually reduced on one side of the square cylinder. This causes the roll-up of the secondary vortex and the increase of the rebound height of the vortex system.

F1.00063 Density measurements in water using background oriented schlieren technique[1] , SHI QIU, CHUANXI WANG, VERONICA ELIASSON, University of Southern California — Undersea earthquakes, tsunamis and underwater explosions are examples of phenomena that cause compressible wave propagation in oceans leading to changes in density and pressure. Here, a direct impact method is used to generate a shock wave in a water-filled channel and the following changes in the density of the fluid is quantified using an extended background oriented schlieren technique. Background oriented schlieren technique relies on measuring variations in index of refraction in the fluid. A high-speed camera is used to capture multiple frames of the shock wave propagation. A code has been developed to quantify the change in index of refection, and map it to the change in density. Results of density changes due to shock wave propagation in converging water-filled channels will be presented.

[1]Supported by Office of Naval Research through a MURI Grant Number N00014-06-1-0730 (Dr. Y.D.S. Rajapakse, Program Manager).

F1.00064 Thermal-Stability Characterization of Quantum Dot on Anodized Aluminum as Global Temperature Sensing , HIROTAKA SAKAUE, JAXA, AKIHISA AIKAWA, Kyushu University — Quantum dot (QD) is an attractive chemical as a temperature-sensitive probe for global temperature sensor. It is bright and has a narrow spectrum compared to those of conventional temperature-sensitive probes. Previous studies show that QD on anodized-aluminum support is applied to high temperature measurement (above 400 K) in hypersonic flows. However, it is also reported that QD is not stable at this temperature range. To answer to the stability issue, we characterize a thermal stability of a QD on anodized-aluminum support. The luminescence from the resultant sensor is measured in the temperature range of 100 to 475 K and the time range of 0 to 1000 s. It is shown that the thermal stability is hold below 298 K. Above 315 K, a sudden decrease and a recovery of the luminescence are measured. It is found that the amount of decrease is proportional to the temperature. The maximum decrease in the intensity is 89% at 475 K after 1000 s. At 315 K, the intensity is recovered to the initial amount after 1000 s.

F1.00065 Effect of PTFE Particle on Super-Hydrophobic Coating for Anti-Icing , RIHO KAMADA, Chuo University, KATSUAKI MORITA, KOJI OKAMOTO, The University of Tokyo, AKIHITO AOKI, SHIGEO KIMURA, Kanagawa Institute of Technology, HIROTAKA SAKAUE, JAXA — Anti/deicing of an aircraft is necessary for a safe flight operation. Mechanical processes, such as heating and deicer boot, are widely used. Deicing fluids should be coated every time before the take-off, since the fluids come off from the aircraft while cruising. We study a super-hydrophobic coating as anti-icing for an aircraft. It is designed to coat the aircraft without removal. Since a super-hydrophobic surface prevents water by reducing the surface energy, it would be another way to prevent ice on the aircraft. We provide a temperature-controlled room, which can control its temperature at the icing conditions (-14 to 0 degrees C). The contact and sliding angles are measured to study the effect of the various PTFE particles on the super-hydrophobic coatings for anti-icing. The particle diameter is varied from 5 to 30 micrometer. Comparisons among the super-hydrophobic coatings by various PTFE particles are made to discuss the performance of the resultant coatings as anti-icer.

F1.00066 Design patterns for training fluid dynamics experimentalists , RANDALL TAGG, University of Colorado Denver - Physics — What practical knowledge would your ideal lab student, technician or even postdoc possess? Borrowing the idea of design patterns from the fields of architecture and computer science, we claim that there are technical problems common to many investigations with often-used design solutions. These would form a useful repertoire that thoughtful practitioners can adapt. Creatively breaking the rules is also encouraged. We invite other ideas for fluid experiment design patterns towards the end of creating a web-based resource that helps new experimentalists get up to speed quickly. We are creating such a resource so that it also serves pre-college students invited into research experiences and inventive citizen scientists exploring new technologies.

F1.00067 ABSTRACT WITHDRAWN —

F1.00068 ABSTRACT WITHDRAWN —

F1.00069 Dynamics of spheroid particles in channel flow , WENBIN MAO, ALEXANDER ALEXEEV, Georgia Institute of Technology, Atlanta, GA, 30332, USA — The effect of inertia on the dynamics of rigid spheroid microparticles in a pressure-driven channel flow is studied using a hybrid lattice Boltzmann and lattice spring method. We find distinctive behaviors of particles depending on the particle shape, initial orientation, and ratio of particle size to the channel size. Two possible stable modes of motion are found for prolate spheroids. Particles either tumble in a shear plane or spin with the axis parallel to the vortex direction. We present a phase diagram showing the transition between these two modes. Cross-stream migration and equilibrium trajectories of particles are also investigated and found to depend on the particle shape and mode of motion. The simulations results are compared with experimental data showing favorable agreement. Our results will be useful for separating biological and synthetic particles by size and shape.

F1.00070 Two-Dimensional Particle-in-Cell Simulation of Cylindrical Magnetron Sputters for the Improvement of Target Utilization

, HUR MIN-YOUNG, BAE HOWN-WON, LEE HO-JUN, LEE HAE-JUN, Pusan Natiomal University — Magnetron sputtering has been commonly used for the deposition of a wide range of industrial thin film coating. This method has almost no restrictions in the target materials and the magnetic field enables lower pressures operation. Conventional flat type sputters generate ion-bombardment sputtering in only a localized region of target where electric fields and magnetic fields are perpendicular to each other. Therefore, the utilization efficiency of the target material is about 20∼30%. To overcome this drawback, a rotating cylindrical target is devised to make uniform sputtering on the target. In this paper, the difference of the physical effects of ions on the targets between the flat-type and the cylindrical-type sputters is investigated using a two-dimensional particle-in-cell simulation with Monte Carlo collisions. Especially for the calculation of cylindrical field solver with a finite difference method, an image charge method is introduced instead of solving the Poisson equation. Simulation Diagnostics include the plasma density, the distributions of energy and angle of incident ions on the target, and the deposition profiles on the substrate calculated by a ray-trace particle deposition model. The analysis for the rotating speed and the magnet structure is to be discussed.

F1.00071 Eulerian and Lagrangian Analysis of a Simulated unsteady Flow Behind a Circular cylinder[1]

, KEITH HOLMES, MELISSA GREEN, Syracuse University — The unsteady wake behind a circular cylinder is numerically simulated and analyzed utilizing various Eulerian methods and the Lagrangian finite-time Lyapunov exponent (FTLE). The objective is to identify and distinguish among shed vortices in the wake, and particular attention is given to the near wake immediately downstream of the cylinder. The Eulerian methods capably determine regions of the flow associated with greatest magnitude of vorticity in the near wake, as has been shown previously. Ridges of the FTLE field are able to objectively identify structures in the near wake by outlining the boundaries between vortices. This includes boundaries among structures of the same sign, a distinction not possible using the Eulerian methods. The formation of these boundaries help to distinguish between structures still developing around the cylinder surface and those that have been shed from the cylinder.

[1]Syracuse University

F1.00072 Physics Based Compressive Sensing Approach Applied to Airfoil Data Collection and Analysis[1]

, ZHE BAI, THAKSHILA WIMALAJEEWA, ZACHARY BERGER, MARK GLAUSER, PRAMOD VARSHNEY, Syracuse University, DONALD LESKIW, Leskiw Associates, SYRACUSE UNIVERSITY TEAM, LESKIW ASSOCIATES TEAM — Compressive Sensing (CS), a newly developing method in signal processing, was used in physics to estimate a two-dimensional, high Reynolds number turbulence flow velocity field over a NACA-4412 airfoil. The facility and experimental setup in the wind tunnel were introduced, from which the velocity data was obtained. The principle of CS and its feasibility applied to the physical velocity field was demonstrated and Discrete Cosine Transform (DCT) was used to attain the sparsity. The reconstructed velocity field was provided to compare with the experimental measurement of the PIV setup and the normalized MSE between the original velocity filed and the reconstructed one was calculated for a number of snapshots, with the comparison of snapshot POD method. Also, a joint CS and POD technique was discussed, in which snapshot POD was used as a basis to find transformations that sparsify the data for CS to retrieve. The fusion of several snapshots was discussed by doing the before and after the CS process. The effectiveness of CS used for the approximation of large and distributed airfoil data sets through a small number of samples collection was demonstrated, which could also be expected to be readily applied to other types of datasets.

[1]1STTR Phase I sponsored by AFOSR

F1.00073 Simulations of bubble-wall collision and bouncing[1]

, SOHRAB TOWFIGHI, HADI MEHRABIAN, Department of Chemical and Biological Engineering, University of British Columbia, ROBERTO ZENIT, Materials Research Institute, National Autonomous University of Mexico, JAMES J. FENG[2], Department of Chemical and Biological Engineering, University of British Columbia — The collision of a rising bubble with a hydrophilic upper wall is studied numerically using an axisymmetric phase-field model. Prior experiments show bubble bouncing or adhesion depending on its incoming velocity. Using experimental parameters, our computation reproduces these different behaviors, including bubble breakup, arrest, and rebound. In particular, dimples are observed on both the fore and aft sides. We further investigate the scaling of the coefficient of restitution and the critical condition delineating arrest from rebound. The latter is plotted as a phase diagram in terms of the Ohnesorge and Weber numbers.

[1]We acknowledge financial support from NSERC
[2]Department of Mathematics, University of British Columbia

F1.00074 Laminar separation bubble formation using a circular cylinder rotating adjacent to a flat plate[1]

, FARHANA AFROZ, AMY LANG, EMILY JONES, MICHAEL BRADSHAW, The University of Alabama — A new method for the formation of a laminar separation bubble (LSB) on a flat plate was investigated whereby an adverse pressure gradient (APG) was induced by the presence of a rotating cylinder. A parametric study was performed where the rotation rate and gap height (G) was changed to vary the strength of APG which affects the nature and extent of the LSB. Results showed that the height (H), length (L) and the separation point (S) of the LSB varied in conjunction with the strength of the APG and the Re. Time-Resolved Digital Particle Image Velocimetry (TR-DPIV) was used to document the LSB in this water tunnel study. Results captured the effects of changing flow speed, cylinder location and rotation rate on the development of the LSB.

[1]Funding under NSF grant 0932352 is gratefully acknowledged.

F1.00075 Selective interaction between microbubbles and modulating waves in a Taylor-Couette flow

, TOMOAKI WATAMURA, YUJI TASAKA, YUICHI MURAI, Hokkaido University — Modifications of a coherent vortical structure by dispersed microbubbles have been investigated in a vertical Taylor-Couette flow, which is the flow generated between coaxial-rotating double cylinders. Radii of the inner and outer cylinders are 95 mm and 105 mm, respectively. The radius ratio and aspect ratio are 0.905 and 20, respectively. Flow mode in the experiments represents wavy vortex flow and modulated wavy vortex flow. Hydrogen bubbles with 60 μm in the mean diameter were generated by water electrolysis and dispersed from a platinum-wire electrode mounted at the bottom of the fluid layer. Maximum void fraction estimated by input power is smaller than 0.01%. Velocity distribution of microbubbles in a Taylor vortex array is determined by image analysis, and show preferential distribution and motion in the oscillating vortex tube. The fluctuation power of the basic wave was increased by adding microbubbles, while the power of its modulation was decreased. The gradient of the azimuthal velocity in the radial direction, i.e. origin of skin frictional drag acting on the cylinder walls, was decreased. These modifications of flow structure represent the suppression of the flow transition, due to the excitation of the basic wave oscillation and increase of momentum transfer by bubble swarm.

F1.00076 Wave localization of linear gravity waves in shallow water: Global measurements and agreement between random matrix theory and experiments[1]

, ANDREA SCHMESSANE, Departamento de Fisica FCFM Universidad de chile, LABORATORY OF MATTER OUT EQUILIBRIUM TEAM[2] — Wave localization explains how a perturbation is trapped by the randomness present in a propagation medium. As it propagates, the localized wave amplitude decreases strongly by multiple internal reflections with randomly positioned scatterers, effectively trapping the perturbation inside the random region. The characteristic length where a localized wave is propagated before being extinguish by randomness is called localization length. We carried experiments in a quasi-onedimensional channel with random bottom in a shallow water regime for surface gravity water waves, using a Perfilometry Fourier Transform method, which enables us to obtain global surface measurements. We discuss keys aspects of the control of variables, the experimental setup and the implementation of the measurement method. Thus, we can control, measure and evaluate fundamental variables present in the localization phenomenon such as the type of randomness, scattering intensity and sample length, which allows us to characterize wave localization. We use the scattering matrix method to compare the experimental measurements with theoretical and numerical predictions, using the Lyapunov exponent of the scattering matrix, and discuss their agreement.

[1]Conicyt

[2]I work in the laboratory of matter out of equilibrium

F1.00077 STABILITY –

F1.00078 The effect of a magnetic field on vortex breakdown in an enclosed swirling flow[1]

, YANG YU, Institute of Thermodynamics and Fluid Mechanics, Ilmenau University of Technology, Ilmenau, Germany, BENWEN LI, Northeastern University, China, ANDRE THESS, Ilmenau University of Technology, Germany — An axisymmetric swirling flow, which is driven by rotation of top lid of an enclosed cylinder and subjected to of an axial uniform magnetic field, is studied numerically. As Reynolds numbers increasing, the phenomenon of vortex breakdown, a vortex appears and disappear on the axis of the cylinder, is a significantly process in the transition from the laminar flow to turbulent flow. The collocation spectral solver is developed to simulate the MHD flow. While, different conductivities of the walls, insulating and perfectly conducting cases, are considered to analyze the magnetic effect on the vortex breakdown. In order to validate the collocation spectral solver, the dynamic and MHD problems are referred, respectively. In the presence of an axial uniform magnetic field, the effects on the non-dimensional lengths of vortex along the z-axis and the central positions of vortex on the z-axis are presented, and the influence of the conductivities of the top lid, bottom base and side wall are discussed. The results show that, for different electrical boundary conditions, the behaviors of vortex are significantly different and even converse. In particular, when the top rotating lid is the only perfectly conducting, the magnetic effects are the strongest.

[1]Research Training Group "Lorentz force velocimetry and Lorentz force eddy current testing" at Ilmenau University of Technology

F1.00079 Three–dimensional Instability in the Wake of an Angulated Cylinder[1]

, KYUNG-SOO YANG, CHOON-BUM CHOI, Inha University — Floquet stability analysis has been carried out to detect the onset of 3D instability in the laminar flow past a 2D angulated cylinder. The shape of the cylinder cross-section considered here includes a normal flat plate up to a square cylinder as the aspect ratio (AR) varies. As AR decreases, mode B and mode QP vanish and mode A2 and mode QP2 emerge. These modes were not observed in the case of flow past a square cylinder. In a finite range of AR, mode A instability becomes unstable and then returns to be stable again as Re increases. This distinctive phenomenon can be identified by a closed curve in the neutral stability diagram. Three–dimensional simulations of some selected cases were performed for validation, showing good agreement with the current Floquet stability analysis. We also present contours of the streamwise vorticity component obtained from the Floquet analysis, and Q contours based on the DNS to elucidate the 3D vortical structures of the 3D modes Our results shed light on a complete understanding of the onset of 3D instability in the presence of an angulated cylinder

[1]This work was supported by the Underwater Vehicle Research Center in Korea.

F1.00080 Experimental Investigation of Richtmyer-Meshkov Instability on Inclined Interface

, CHRIS MCDONALD, JACOB MCFARLAND, Texas A&M University, DAVID REILEY, University of North Texas, BRIAN REID, DEVESH RANJAN, Texas A&M University — Results are presented from our recent experiments studying shock wave interaction with an inclined interface between two different fluids performed in a new newly built Texas A&M variable inclination shock tube facility. The variable inclination capability of the shock tube allows for an inclined interface to be created with ease, without changing the Mach number (pressure gradient) or Atwood number (density gradient). The ease of creating the interface provides a clean and repeatable interface for studying the Richtmyer-Meshkov Instability problem. The results presented from our initial experiments are from a Mach 1.6 shock wave interaction with a nitrogen-infused-fog-over-carbon dioxide interface for an inclination angle of 60 degrees. Quantitative results gathered from these experiments such as the mixing width growth rate, and vorticity deposition will be discussed in detail. Numerical simulations of the experiments are performed using the ARES code (LLNL) and the time evolution of the interface width, measured empirically, is compared to the corresponding numerical predictions.

F1.00081 Spatiotemporal spectral analysis of a forced cylinder wake[1]

, JUAN D'ADAMO, Facultad de Ingeniería, Universidad de Buenos Aires (CONICET), RAMIRO GODOY-DIANA, JOSÉ EDUARDO WESFREID, PMMH UMR7636 CNRS, ESPCI ParisTech, UPMC (Paris 6), U. Paris Diderot (Paris 7) — The wake of a circular cylinder performing rotary oscillations is studied using hydrodynamic tunnel experiments at $Re = 100$. Two-dimensional particle image velocimetry on the mid-plane perpendicular to the axis of cylinder is used to characterize the spatial development of the flow and its stability properties. The lock-in phenomenon that determines the boundaries between regions of the forcing parameter space were the wake is globally unstable or convectively unstable is scrutinized using this experimental data. A novel method based on the analysis of power density spectra of the flow allows us to give a detailed description of the forced wake, shedding light on the energy distribution in the different frequency components and in particular on a cascade-like mechanism evidenced for a high amplitude of the forcing oscillation. In addition, a calculation of the drag from the velocity field is performed, allowing us to relate the resulting force on the body to the wake properties. Reference: Phys. Rev. E 84, 056308 (2011).

[1]The present work was supported by the Franco-Argentinian Associated Laboratory in the Physics and Mechanics of Fluids (LIA PMF-FMF).

F1.00082 Analysis of Görtler Vortices Spanwise Wavelenght Influence in Heat Transfer Rates

, LEANDRO F. SOUZA, VINICIUS MALATESTA, University of Sao Paulo, JOSEPH T.C. LIU, Brown University — The centrifugal instability mechanism in boundary layers flows over concave surfaces is responsible for the development of streamwise counter-rotating vortices, known as Görtler vortices. These Vortices create two regions in the spanwise direction, the upwash and downwash regions. The downwash region is responsible to compress the boundary layer towards the wall, increasing drag and heat transfer rates. The upwash region does the opposite. In the nonlinear development of the Görtler vortices the upwash region becomes narrow, and the average drag and heat transfer rate is higher than that for a Blasius boundary layer. In the present research, using a Spatial Direct Numerical Simulation, it is analyzed the influence of the Görtler Vortices spanwise wavelength in heat transfer rates. Different wavelengths are analyzed and compared with experiments.[1] The results show that steady Görtler flow can reach heat transfer rates higher than the turbulent values, even without introducing secondary instabilities.

[1]L. Momayez, P. Dupont and H. Peerhossaini, Int J Therm Sci, 43, 753–760 (2004)

F1.00083 Flow Control Characterization of PIV Flow Field Using POD[1] , MATTHEW BERRY, ZACH BERGER, KERWIN LOW, ALEXIS ZELENYAK, Syracuse, SIVARAM GOGINENI, Spectral Energies LLC , MARK GLAUSER, Syracuse — The main focus of this experiment is on the analysis and flow control of a high speed jet, with a nozzle diameter 2". The flow field was investigated at Mach 0.6 using two-component large window PIV. The velocity was examined in the streamwise direction of the r-z plane with a window size of about 6 diameters. A glove fitted with 8 synthetic actuators was attached and arranged azimuthally around the lip of the jet. A signal was driven through the actuation system causing the glove to inject flow into the jet field in an attempt to clean up the jets shear layer. Sensors were set up in order to sample the near field pressure measurements and the far field acoustics simultaneously with the PIV. Low-dimensional modelling was performed on the gloves actuation system running alone; without the high speed jet. Proper orthogonal decomposition was performed at the lip of the jet, in order to observe the high energy structures of the actuation glove. Previous work shows that the first mode consists of 15% of the total energy, while the first 9 modes are responsible for approximately 40% of the total energy. Combing the PIV information with a complete hotwire velocity profile, we can gain a better understanding of how the closed loop control system is affecting the jets flow field.

[1]Syracuse University and Spectral Energies LLC

F1.00084 Magnetic resonance imaging study of Rayleigh-Benard convection in near and supercritical hexafluoroethane , JOSHUA M. BRAY, SARAH L. CODD, Mechanical and Industrial Engineering, Montana State University, JOSEPH D. SEYMOUR, Chemical and Biological Engineering, Montana State University — Rayleigh-Benard convection (RBC) has been extensively studied; however, the regime of temperature and pressure above the critical point (Tc, Pc) remains largely unexplored experimentally. In this regime, convection modes are sensitive to divergence in various transport properties, providing a unique model system for many geophysical flows and allowing access to high Rayleigh numbers Ra, of order 10^{13}. We present magnetic resonance imaging (MRI) analysis of RBC in a supercritical fluid. Spatially resolved velocity images and ensemble-averaged transport dynamics were acquired non-invasively for C_2F_6 (Tc = 20 C, Pc = 31 bar) in a low aspect ratio chamber at pressures 32 bar and 70 bar with and without a 2.5 C temperature gradient. With no applied gradient, within the temperature control resolution of +/-0.1 C, convection at 70 bar is minimal, but is rapid at 32 bar due to near-critical density divergence. The velocity profile is concentric, in contrast to a single convective cell in non-critical fluids. An applied temperature gradient produced turbulent flow, destabilizing the concentric velocity distribution at 70 bar and generating incoherent fluid motion at 32 bar. Near and supercritical fluids provide a model system for tuning fluid dynamics through thermodynamics.

F1.00085 Using Lyapunov Vectors to Quantify Spatiotemporal Chaos in Rayleigh-Benard Convection , MU XU, Virginia Tech, ALIREZA KARIMI, University of Notre Dame, JEFFREY TITHOF, Georgia Institute of Technology, MIRO KRAMAR, VIDIT NANDA, Rutgers University, MICHAEL SCHATZ, Georgia Institute of Technology, KONSTANTIN MISCHAIKOW, Rutgers University, MARK PAUL, Virginia Tech — Spatiotemporal chaos is a common feature of spatially-extended systems that are driven far-from-equilibrium with examples that include the dynamics of the weather and climate, fluid turbulence, and excitable media. Despite significant effort, many open questions remain regarding our physical understanding of high-dimensional chaotic systems such as these. Using recent advances in computing algorithms and available supercomputing resources it is now possible to compute the spectrum of Lyapunov exponents and orthonormal Lyapunov vectors for experimental conditions. We present large-scale parallel numerical simulations of Rayleigh-Benard convection undergoing spiral defect chaos for very long times and for laboratory conditions that we compare with experiment. We use averages of the leading order orthonormal Lyapunov vector to gain insight into the regions in space generating the most disorder which we compare with experimentally accessible quantities. We discuss the similarities and differences between characteristic and orthonormal Lyapunov vectors. Lastly, we describe our efforts to compute the characteristic Lyapunov vectors for Rayleigh-Benard convection.

F1.00086 Penetrative convection induced by a statically unstable density distribution in a very thin central layer , RISHAD SHAHMUROV, LAYACHI HADJI, The University of Alabama — Several models of penetrative convection have been studied (Gribov & Gurevich, 1957; Veronis, 1963; Matthews, 1988; Batchelor & Nitsche, 1990; Simitev & Busse, 2010). We consider Rayleigh-Bénard convection with a static density distribution that has a piecewise linear dependence on the vertical coordinate and whose unstably stratified part occupies a central layer of thickness $\epsilon \ll 1$. Some limiting cases corresponding to the linear eigenvalue problem are treated analytically and the results confirmed by a detailed numerical investigation. Steady two-dimensional flow patterns are determined numerically for supercritical Rayleigh numbers in the range $\epsilon \geq 0.06$. For $0.2 \leq \epsilon \leq 0.5$, an analytical nonlinear stability three-dimensional study is undertaken in the case of poorly conducting boundaries. A weakly nonlinear evolution equation for the leading order temperature perturbation is also derived and solved numerically as function of ϵ and Prandtl number. The effect of the boundaries on the flow characteristics diminishes as $\epsilon \to 0$, leading us to study the stability of an unbounded stratified fluid for which similarity type solutions are obtained. Our findings are compared to those of the models mentioned above.

F1.00087 Investigation of instability of displacement front in non-isothermal flow problems[1] , NATALIA SYULYUKINA, Lomonosov Moscow State University, ANNA PERGAMENT, Keldysh Institute of Applied Mathematics, Russian Academy of Sciences — In this paper, we investigate the issues of front instability arising in non-isothermal flow displacement processes. The problem of two-phase flow of immiscible fluids, oil and water, is considered, including sources and dependence of viscosity on temperature. Three-dimensional problem with perturbation close to the injection well was considered to find the characteristic scale of the instability. As a result of numerical calculations, theoretical studies on the development of the instability due to the fact that the viscosity of the displacing fluid is less than the viscosity of the displaced have been confirmed. The influence of temperature on the evolution of the instability was considered. For this purpose, the dependence of oil viscosity on temperature has been added to the problem. Numerical calculations were carried out for different values of temperature and it was shown that with increasing of production rate. Thus, it has been demonstrated that the selection of the optimal temperature for injected fluids a possible way for stimulation of oil production also delaying the field water-flooding.

[1]This work was supporting by the RFBR grant 12-01-00793-a.

F1.00088 Understanding the impact of initial condition on low Atwood number Rayleigh-Taylor driven flows , SARAT CHANDRA KUCHIBHATLA, DEVESH RANJAN, Texas A&M University — Experimental investigation of the effects of initial conditions on Rayleigh-Taylor instability was performed using the Water Channel facility at Texas A&M University. Hot and cold water (with a temperature difference of ∼7-8 degrees C) selected as working fluids were unstably stratified using a splitter plate resulting in a low Atwood number of ∼0.0015. Using a servo controlled flapper system the effect of initial conditions is studied using different diagnostics such as optical imaging, thermocouples and hot-wire anemometry. A parametric study comprising of up to 10 modes of the initial condition was performed by varying the number of modes as well as modal composition ($i.e.$ ratio of wavenumbers and phase differences). Variation of density, temperature and velocity field in the linear and non-linear stages of RT growth was recorded and analyzed. At non-dimensional time, $t^* = t(A_t g/H)^{0.5} = 1.3$, where t is the time, H is the width of the Channel, and g is the acceleration due to gravity, power spectra of the non-dimensional density showed fine-scale components that are dependent upon the initial condition. Plots of scalar dissipation and mixing rate indicate greater dissipation rate at early times that tends to asymptote to the order of kinematic viscosity at late times.

F1.00089 Miscible and immiscible experiments on the Rayleigh-Taylor instability using simultaneous planar laser induced fluorescence and backlight 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 motors and accelerated downward causing the initially stable interface to be unstable and allowing the Rayleigh-Taylor instability to develop. Experiments are presented with and without forced initial perturbations produced by vertically oscillating the test sled prior to the start of acceleration. Half of the experimental tank is visualized using a 445nm laser light source that illuminates a fluorescent dye mixed in one of the fluids. The other half is illuminated with a white backlight. The resulting images are recorded using a monochromatic high speed video camera allowing for the measurement of spike and bubble mixing layer growth rates for both visualization techniques in a single experiment.

F1.00090 Simulations of MHD Instabilities in Protoplanetary Disks , EZEKIEL HADLEY, JOSEPH BARRANCO, San Francisco State University — We have developed a 3D spectral, anelastic, magnetohydrodynamic code for rapidly rotating, strongly sheared, stratified protoplanetary disks. The Cartesian domain is resolved with a Fourier-Fourier-Chebyshev basis, and uses horizontal coordinates that advect with the background shear. With this tool, we simulate the magnetorotational instability at high resolution and investigate the development of MHD turbulence.

F1.00091 TURBULENCE —

F1.00092 Passive and active transport in a chaotic flow field , CHRISTOPHER MEHRVARZI, MARK PAUL, Virginia Tech — The transport of a scalar species in a complex flow field is important in many areas of current interest such as the combustion of premixed gases, the dynamics of particles in the atmosphere and oceans, and the reaction of chemicals in a mixture. There has been significant progress in understanding transport in steady periodic flows such as a ring of vortices. In addition, transport in turbulent flow has an extensive literature. However, in this work, we focus upon the transport of a scalar species in a three-dimensional time-dependent flow field given by the spiral defect chaos state of Rayleigh-Benard convection (the buoyant convection that results when a shallow fluid layer is heated from below). We take advantage of the significant theoretical and numerical progress in recent years that provides a physical understanding of this chaotic flow field. 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 cylindrical domains with experimentally relevant boundary conditions. For active transport we include a reaction term with relevance to the combustion of pre-mixed gases that are undergoing chaotic convection. We develop and use diagnostic tools to quantify the transport over a wide range of parameters in order to gain new physical insights.

F1.00093 Turbulent Heat Transfer in Ribbed Pipe Flow[1] , CHANGWOO KANG, KYUNG-SOO YANG, Inha University — From the view point of heat transfer control, surface roughness is one of the popular ways adopted for enhancing heat transfer in turbulent pipe flow. Such a surface roughness is often modeled with a rib. In the current investigation, Large Eddy Simulation has been performed for turbulent flow in a pipe with periodically-mounted ribs at Re_τ=700, Pr=0.71, and p/k=2, 4, and 8. Here, p and k represent the pitch and rib height, respectively. The rib height is fixed as one tenth of the pipe radius. The profiles of mean velocity components, mean temperature, root-mean-squares (rms) of temperature fluctuation are presented at the selected streamwise locations. In comparison with the smooth-pipe case at the same *Re* and *Pr*, the effects of the ribs are clearly identified, leading to overall enhancement of turbulent heat transfer in terms of *Nu*. The budget of temperature variance is presented in the form of contours. The results of an Octant analysis are also given to elucidate the dominant events. Our LES results shed light on a complete understanding of the heat-transfer mechanisms in turbulent ribbed-pipe flow which has numerous applications in engineering.

[1]This work was supported by the National Research Foundation of Korea (NRF) grant funded by the Korea government (MEST) (No. 2012013019).

F1.00094 Experimental Analysis of Coherent Structures in the Unsteady Wake of a Circular Cylinder , JACOB MORRIDA, Syracuse University — An experimental analysis of the two-dimensional unsteady wake behind a circular cylinder was studied at a range of Reynolds number, and the data was compared with numerical results. The near wake region was analyzed to better understand the evolution of coherent structures, in pursuit of effective control over when and where vortex shedding occurs. For the experiments, a DPIV measurement system was used to collect two-dimensional velocity data. The properties of the wake were determined by Eulerian vortex criteria and Lagrangian coherent structures. Using these methods of analysis, the regions of flow separation and vortex shedding were closely observed to discover when and where the separation occurred. Of particular interest was whether identification of this shedding using the two forms of analysis differed between the experimental and numerical results. The ultimate goal is to objectively identify the shedding phenomenon as a target for flow control in future applications.

F1.00095 Geostrophic balance and the emergence of helicity in rotating stratified turbulence , RAFFAELE MARINO, PABLO D. MININNI, DUANE ROSENBERG, ANNICK POUQUET, Institute for Mathematics Applied to Geosciences (IMAGe), CISL/NCAR, P.O. Box 3000, Boulder, Colorado 80307-3000, USA — We perform numerical simulations of decaying rotating stratified turbulence and show, in the Boussinesq framework, that helicity (velocity-vorticity correlation), as observed in super-cell storms and hurricanes, is created due to geostrophic balance common to large-scale atmospheric and oceanic flows. Helicity emerges from the joint action of eddies and of inertial and gravity waves of respective frequencies f and N, and it occurs when the waves are sufficiently strong, with $N/f < 3$. Outside this regime, and up to the highest Reynolds number obtained in this study, namely $Re \approx 10^4$, helicity production is found to be persistent for N/f as large as ~ 17.

F1.00096 Development and assessment of dynamic water-surface roughness model for large-eddy simulation of winds blowing over water waves[1] , LIAN SHEN, DI YANG, Civil Engineering, Johns Hopkins University, CHARLES MENEVEAU, Mechanical Engineering & CEAFM, Johns Hopkins University — Large-eddy simulation (LES) has become a useful tool for the study of turbulent winds blowing over water surfaces. The surfaces are usually covered by waves with various lengths, interacting with the wind turbulence over a wide range of scales. In the LES, in addition to the subgrid-scale (SGS) stress in the flow, the SGS surface roughness also needs to be modeled. In this study, following the work by Anderson and Meneveau (JFM, 2010) on the dynamic modeling of the SGS roughness of stationary surfaces, we have developed a new dynamic model for the water-surface SGS roughness based on the physics of surface waves. The roughness is quantified by an integral of the SGS wave spectrum weighted based on the wind-wave kinematics, with an unknown model coefficient as prefactor. This coefficient is determined dynamically based on the first-principles constraint that the total surface drag force must be independent of the LES filter scale. This new roughness model is assessed by *a priori* and *a posteriori* tests, and is found to successfully capture the effects of SGS surface waves on the wind turbulence without *ad hoc* prescription of model parameters.

[1]LS and DY acknowledge the support of ONR-N00014-09-1-0395. CM acknowledges the support of NSF-AGS-1045189.

F1.00097 Generation of hairpin vortex packet in channel flow at $Re_\tau \leq 590$[1] , KYOUNGYOUN KIM, Hanbat National University, South Korea — The generation of hairpin vortex packet from an initial single vortex is examined by direct numerical simulation for channel flows at Re_τ = 180, 395, and 590. The initial vortex structure is given by conditionally averaged flow field with the Q2 event specified at $y^+ = 50$ in fully developed turbulent channel flow. The vortex packet formation in higher Reynolds number flows is very similar to that in Re_τ =180 reported by Zhou *et al.* (1999, J. Fluid Mech.); the initial vortex is developed to a primary hairpin vortex (PHV) and the secondary hairpin vortex is generated upstream of PHV. As time proceeds, the vortices move downstream with almost the same convection velocity and little dispersion, forming a vortex packet. Comparison of the packet formation for different Re_τ reveals that the secondary hairpin vortex is generated with time scales based on the wall units. At the time when the head of PHV has grown to the channel center, the inclination angle of the vortex packet is approximately $12 \sim 14^\circ$ which is insensitive to Re_τ, consistently with linear stochastic estimation results with PIV measurement by Christensen & Adrian (2001, J. Fluid Mech.).

[1]Supported by NRF Grant 2010-0007901.

F1.00098 Effect of Seeding Particles on the Shock Structure of a Supersonic Jet[1] , DAVID PORTA, Facultad de Ciencias UNAM, CARLOS ECHEVERRÍA, CATALINA STERN, Facultad de Ciencias, UNAM — The original goal of our work was to measure. With PIV, the velocity field of a supersonic flow produced by the discharge of air through a 4mm cylindrical nozzle. The results were superposed to a shadowgraph and combined with previous density measurements made with a Rayleigh scattering technique. The idea was to see if there were any changes in the flow field, close to the high density areas near the shocks. Shadowgraphs were made with and without seeding particles, (spheres of titanium dioxide). Surprisingly, it was observed that the flow structure with particles was shifted in the direction opposite to the flow with respect to the flow structure obtained without seeds. This result might contradict the belief that the seeding particles do not affect the flow and that the speed of the seeds correspond to the local speed of the flow.

[1]We acknowledge support from DGAPA UNAM through project IN117712 and from Facultad de Ciencias UNAM

F1.00099 Calibration of Discrete Random Walk (DRW) Model via G.I Taylor's Dispersion Theory[1] , TEYMOUR JAVAHERCHI, ALBERTO ALISEDA, University of Washington — Prediction of particle dispersion in turbulent flows is still an important challenge with many applications to environmental, as well as industrial, fluid mechanics. Several models of dispersion have been developed to predict particle trajectories and their relative velocities, in combination with a RANS-based simulation of the background flow. The interaction of the particles with the velocity fluctuations at different turbulent scales represents a significant difficulty in generalizing the models to the wide range of flows where they are used. We focus our attention on the Discrete Random Walk (DRW) model applied to flow in a channel, particularly to the selection of eddies lifetimes as realizations of a Poisson distribution with a mean value proportional to κ/ϵ. We present a general method to determine the constant of this proportionality by matching the DRW model dispersion predictions for fluid element and particle dispersion to G.I Taylor's classical dispersion theory. This model parameter is critical to the magnitude of predicted dispersion. A case study of its influence on sedimentation of suspended particles in a tidal channel with an array of Marine Hydrokinetic (MHK) turbines highlights the dependency of results on this time scale parameter.

[1]Support from US DOE through the Northwest National Marine Renewable Energy Center, a UW-OSU partnership.

F1.00100 A Comparative Study of Spatially Evolving Self-Propelled Wakes and a Patch of Turbulence in a Stratified Fluid , ANIKESH PAL, MATTHEW DE STADLER, SUTANU SARKAR, University of California San Diego — Direct numerical simulation (DNS) is used to compare the evolution of a self-propelled wake and a patch of turbulence in a stratified fluid. The primary focus of this study is to determine the influence of the mean velocity profile on the evolution of the wake. The cases considered are: (a) self-propelled wake with a canonical mean velocity profile, (b) self-propelled wake with the mean removed, (c) a patch of turbulence with the same initial energy spectrum, and (d) a patch of turbulence with a different initial energy spectrum. The Reynolds number (Re), Froude number (Fr) and Prandtl number (Pr) used in the simulations are 10000, 3.194 and 1 respectively. Auxiliary temporally evolving simulations are used to generate the inlet boundary conditions for the spatially evolving cases. We obtain qualitatively and quantitatively similar results among (a), (b) and (c) for the evolution of kinetic energy, vortical structures and the wave field. However the wake dynamics in case (d) with a different initial energy spectrum show significant differences. The implication is that the fluctuating components of the velocities close to a body determine the subsequent flow. The mean velocity may exert an indirect influence through its influence on the near-wake turbulence spectrum.

F1.00101 Volumetric Lattice Boltzmann Simulation for Fluid dynamics and Turbulence in Practical Syringes , EVERTON LIMA, Department of Computer and Information Science, Department of Mathematical Sciences, Indiana University-Purdue University Indianapolis (IUPUI), DEBANJAN DEEP, HUIDAN (WHITNEY) YU, Department of Mechanical Engineering,Indiana University Purdue University Indianapolis (IUPUI) — We conduct numerical experiments to study fluid dynamics and turbulence in syringes using volumetric lattice Boltzmann method (VLBM) that is developed for dealing with arbitrary moving boundaries. Several common used medical syringes are used to predict the efficiency and safety of syringes experiencing low flow infusion rates. It is found that smaller size syringes reach a steady flow rate much sooner than larger ones, which are in quantitative agreement with experimental results. The relation between the syringe size and its steady flow rate is revealed. At low flow rates, corner vortices are observed. We explore conditions that lead to turbulent flow aiming to aid safer syringe application in nursing practices.

F1.00102 10kHz TRPIV near-field velocity and far-field noise: experimental results of a Mach 0.6 jet[1] , PINQING KAN, JACQUES LEWALLE, Syracuse University — Last year, we extracted footprints of sources from far-field pressure data of high speed jet. In this paper, we focus on processing 10kHz TR-PIV data in the near-field region. The velocity and pressure data were collected in the jet experiment of Kerwin Low, et al in 2011. From the PIV snapshots, we extract kinematic indicators, construct their histories over the sequence of snapshots, and select the most promising diagnostics by cross-correlation with far-field pressure. For these indicators, including mass flux, we map out their fluctuations and their downstream propagation. In particular, the phase velocity of fluctuations is compared to the local convective speed. Events with large relative phase velocity are identified, and the corresponding velocity and vorticity fields are mapped out. Our goal is to correlate such events to the far-field footprints of sources. For more info and results related to the experiment, please refer to abstracts by Z.P. Berger and by M.G. Berry at this meeting. This work is supported by Glauser group of Syracuse University, Spectral Energies LLC under AFOSR SBIR grant, and by a Syracuse University Graduate Fellowship.

[1]Thanks for partial support from Spectral Energies LLC (under SBIR grant from AFOSR), Syracuse University and the LCS College

F1.00103 Comparison of 10kHz TR-PIV and LES near-field data in high speed jets[1] , JACQUES LEWALLE, PINQING KAN, Syracuse University — The identification of the sources of noise in high-speed jets may help formulate control strategies, an important unsolved problem. We report on the existence of large intermittent and localized relative phase velocities for near-jet fluctuations, and on the flow patterns that are associated with them (see companion abstract by P. Kan). Here we analyze two data sets. Experimentally, 10 kHz TR-PIV in a $Ma = 0.6$ cold jet (Re = 700,000) yielded two components of velocity, from which we calculate the phase velocities for various indicators (see related abstracts by Z.P. Berger and by M.G. Berry; data provided by Spectral Energies LLC). Similar results are obtained for Ma=0.9 LES results (Re = 400,000, sampling at 80 kHz). The comparison of algorithms and flow patterns vindicates our approach. Correlations with far-field events will also be attempted. Thanks to Guillaume Daviller (Institut PPrime, France) for the LES data, and to the Glauser group at Syracuse University.

[1]Thanks for partial support from Spectral Energies LLC (under SBIR grant from AFOSR), Syracuse University and the LCS College.

F1.00104 An experimental study of the turbulent development of Richtmyer-Meshkov instability with a random initial perturbation , VLADIMER TSIKLASHVILI, OLEG LOKHATCHEV, JEFFREY JACOBS, University of Arizona — Richtmyer-Meshkov (RM) instability is studied in a vertical shock tube experiment. The instability is observed between two gases of different densities accelerated by an incident planar shock wave. The stable stratification of the gases is created by introducing air seeded with smoke through a plenum at the top of the driven section, and SF6 through a plenum at the bottom. The gases are oscillated vertically using two loud speakers, located at the top and bottom of the driven section. Faraday waves created on the interface of the two gases results in a random initial perturbation from which the RM instability develops. The current study focuses on the development of the turbulent mixing layer width following the shock-interface interaction. In past experiments, a variety of growth behaviors has been observed. In some experiments the mixing layer width initially grows rapidly and then saturates later on. Other experiments have more gradual, almost linear growth behavior. In the new experiments views of the initial perturbation are captured along with the growth behavior in order to determine the effects of initial conditions on the mixing layers width's development.

F1.00105 Fluid dynamics at transition regions of enhanced heat transfer channels[1] , JENNIFER C. CASE, NICHOLAS A. POHLMAN, Northern Illinois University — Helical wire coil inserts are used to enhance heat transfer in high heat flux cooling channels. Past research using temperature probes has sufficiently proven that wire coils increase heat transfer by factors of three to five through the disruption of the boundary layer in the channels. The coils are passive devices that are inexpensive to manufacture and easily integrate into existing heat exchangers given the limited pressure drop they produce. Most of the fluid mechanics research in flow over helical coils has focused on the dynamics and vortex structure in fully developed regions rather than the short transition region where the enhanced heat transfer is often expected. Understanding how the development of the flow occurs over the axial length of the cooling channel will determine minimum dimensions necessary for enhanced heat transfer. Results of particle-shadow velocimetry (PSV) measurements report on the flow velocities and turbulence that occurs in the transition regions at the beginning of wire coil inserts. The ability to relate parameters such as flow rate, wire diameter, coil pitch, and the total tube length will increase fundamental knowledge and will allow for more efficient heat exchanger designs.

[1]Funding provided by NIU's Undergraduate Special Opportunities in Artistry & Research grant program.

F1.00106 Characterization of shear turbulence in Keplerian systems[1] , ZOE YAN, Princeton Plasma Physics Laboratory — The process by which astrophysical Keplerian disks transport angular momentum is not well understood. Cool proto-planetary disks may not be sufficiently ionized for magneto-hydrodynamic forces to assist accretion and prompts the question of whether a hydrodynamic pathway for angular momentum transport is possible. The Hydrodynamic Turbulence Experiment studies turbulence evolution in differentially rotating systems. Ekman effects are mitigated by controlling the axial boundary conditions with rings which rotate independently of the inner and outer cylinders. Azimuthal and radial velocities are measured locally using a Laser Doppler Velocimetry diagnostic. Turbulence decay time-scales and Reynolds stress were inferred from these measurements. The interaction of turbulence and rotation is probed by forcing perturbations from a set of configurable jets mounted on the inner cylinder. In all cases examined, the turbulence dies away exponentially, suggesting that at least for the perturbation amplitudes achieved, no subcritical transition exists in these systems. Lifetimes of turbulent states will be presented as a function of the dimensionless shear.

[1]This project was sponsored by the DOE-SULI program and PPPL.

F1.00107 On Self-Similarity of Turbulent Flows over Porous Media , ARIANE PAPKE, Max Planck Institute for Dynamics and Self-Organization, ILENIA BATTIATO, Clemson University — Coupled flows through and over porous layers occur in a variety of natural phenomena, biological systems and industrial processes. Some examples include turbulent flows over sediment beds, urban canopies, polymer brushes and packed-bed heat exchangers. Though such systems span a broad range of spatial scales, recent experiments [Ghisalberti, 2009] suggest the existence of a self-similar behavior. In this work we employ a two-domain approach to model flow through and over a porous medium: it couples the Reynolds with the Darcy-Brinkman equation, and allows one to derive analytical expressions of relevant quantities, such as interfacial velocity and drag length scale, just to mention a few. The connection between our analytical results and experimental data is discussed.

F1.00108 ENERGY –

F1.00109 Dynamic behavior of low-dimensional model for double diffusive natural convection , YOHSUKE YAMADA, RIYOTA TAKEUCHI, HIROSHI GOTODA, Ritsumeikan University — From the viewpoint of a nonlinear dynamical system, an understanding of the physics in the double diffusive natural convection is crucial in present-day engineering and natural science. We discuss the dynamic behavior of the intermittent chaos region in the double diffusive natural convection produced by a fifth-order nonlinear dynamical system. After the intermittent chaos region become complex with increasing normalized Rayleigh number, it undergoes a significant transition to steady-state through reverse period-doubling bifurcation cascade. The dynamic properties of the phase space are investigated in detail in this presentation, which have not been reported in previous theoretical research on dynamical systems.

F1.00110 Numerical study on heat transfer and flow of natural convection in a square enclosure with two vertically aligned cylinders , YONG GAP PARK, MAN YEONG HA, School of Mechanical Engineering, Pusan National University, Jang Jeon 2-Dong, Geum Jeong Gu, Busan 609-735, Korea, HYUN SIK YOON, Global core research center for ships and offshore plants, Pusan National University, Jang Jeon 2-Dong, Geum Jeong Gu, Busan 609-735, Korea — This study investigates natural convection in a square enclosure with two hot inner cylinders induced by a temperature difference between a cold outer enclosure and two hot inner circular cylinders. The centers of two equidiameter cylinders are placed at those of the lower and upper half of the enclosure, respectively. The immersed boundary method (IBM) to model the inner circular cylinders based on the finite volume method is used to study a two-dimensional natural convection for different Rayleigh numbers varying from 10^3 to 10^6. The effect of the distance between two inner cylinders in an enclosure on heat transfer and fluid flow at different Rayleigh numbers has been examined. The distance between two inner cylinders is changed from 0.1 to 0.5. The natural convection bifurcates from steady to unsteady depending on Ra and the distance between two inner cylinders . The flow and thermal fields eventually reach steady state regardless of the distance between two inner cylinders in the range of Rayleigh numbers from 10^3 to 10^5. However, for Ra = 10^6, there exist unsteady regions depend on the distance between two inner cylinders.

F1.00111 A numerical study of natural convection in a square enclosure with a circular cylinder at different temperature of bottom surface , MINSUNG KIM, MAN YEONG HA, HYUN SIK YOON, Pusan National University — The physical model considered here is a square enclosure of fluid heated below and cold above with a hot cylinder placed at the center of the square. The bottom of square enclosure has dimensionless thermal isothermals of 0 to 1.0. The immersed boundary method (IBM) to model an inner circular cylinder based on finite volume method (FVM) is used to study for different Rayleigh numbers varying over the range of 10^3 $\sim 10^6$. The dimensionless temperature of the bottom wall is changed along the hot cylinder located the center of the square. This study investigates the effects of the temperature of bottom wall and the buoyancy-induced convection on heat transfer and fluid flow. Detailed analysis results for the distribution of streamlines, isotherms and Nusselt number are presented in this paper.

F1.00112 Prediction of thermal hydraulic characteristics inside the storage tank of a horizontal condensation heat exchanger using MARS-KS , BYUNG SOO SHIN, KWANG WON SEUL, KYU SIK DO, Korea Institute of Nuclear Safety, REACTOR SYSTEM EVALUATION TEAM — The performance of a horizontal condensation heat exchanger is determined by the condensation heat transfer inside the heat exchanger tubes, convective or boiling heat transfer outside the tubes and flow characteristics in the storage tank. The flow characteristics in the tank are important factors to determine the heat transfer rate outside the tubes. The objective of this work is to develop the method to predict the heat transfer rate outside the tubes properly using MARS-KS code. Two different results from MARS-KS were compared with simplified experimental results in other works to estimate the capacity of MARS-KS. One was by a typical 1D nodalization but another was by a 3D nodalization considering natural circulation in the storage tank. Then, to eliminate the effect of condensation heat transfer inside the tubes, the experimental results on temperature profiles were applied to the inside wall of tubes as boundary conditions. As the result, the 3-D nodalization model had good predictions with experimental results in regard of wall temperature, heat flux and heat transfer coefficients. It was also confirmed that the natural circulation flow was developed inside the storage tank.

F1.00113 Heat transfer and fluid flow of solid-liquid two-phase media of different heat conductivities , TAKAAKI TSUTSUMI, SHINTARO TAKEUCHI, TAKEO KAJISHIMA, Department of Mechanical Engineering, Osaka University, Japan — A direct numerical simulation of particle-laden flows, which incorporates the effects of temperature gradient within solid object and heat conduction through moving boundaries, is applied to investigate the heat transfer in a dispersed two-phase media. The momentum exchange at the fluid-solid boundaries is treated by our immersed solid approach. A flux-decomposition scheme is proposed for the heat conduction at the interface. Then we developed an implicit scheme which has wide ranges of applicability for solid-fluid density and heat-conductivity ratios, Reynolds number and Rayleigh number. A two-dimensional natural convection of a mixture composed by liquid and dispersed circular particles of neutral density, confined in a square domain, is simulated. Influences of heat-conductivity ratio and volumetric fraction of solid in the liquid are particularly observed. In case of relatively low volume fraction of solid, the scale of circulating flows is dominated by the heat-conductivity ratio. In case of dense concentration of particles, the heat transfer due to inter-particle connections and/or vibratory motions of particles become pronounced. Overall, these findings highlight the importance of temperature distributions within the particles as well as in the liquid.

F1.00114 Modeling thermophoretic deposition of particles from a hot fluid stream , ZACHARY MILLS, WENBIN MAO, Georgia Tech, ALOK WAREY, ANIL SINGH BIKA[1], VENKATESH GOPALAKRISHNAN, General Motors Research and Development Center, ALEXANDER ALEXEEV, Georgia Tech — We developed a three dimensional computational model to examine the deposition of aerosol particles in heat exchangers. Our model combines a thermal lattice Boltzmann model for simulating the fluid flow and temperature distribution in the heat exchanger and a Brownian dynamics model that is used to model the transport and deposition of aerosol particles. In our simulations, we investigated particle deposition resulting from convection, thermophoresis, and diffusion. To validate our model we directly compare the simulation results with experimental data for the deposition of particles in a model heat exchanger. We augment our model with a model that describes particle adhesion to the heat exchanger walls. It allows us to examine the formation and buildup of the deposit layer for different flow conditions and particle distributions. Thus, our results provide useful insights into the deposit formation process that are needed for designing heat exchangers that a less prone to fouling.

[1]University of Minnesota, Minneapolis

F1.00115 Geometric Effects on Power Generation by Reverse Electrodialysiswith Self-induced Electrolyte Flow in Ion-Selective Nanochannels , BYOUNG JAE KIM, Korea Atomic Energy Research Institue, DONG-KWON KIM, Ajou University, SEUNG-HYUN LEE[1], Korea Institute of Machinery & Materials — Recently, solid-state nanofluidic channels or nanopores have been demonstrated experimentally to serve as ion-selective membranes for small reverse electrodialysis systems. Ions of opposite charge to that of the surface (counter-ions) are attracted toward the surface while ions of like charge (co-ions) are repelled from the surface. As a result, the counter-ions are preferentially transported over the co-ions in the charged nanochannels. Under a concentration gradient, the ions diffuse spontaneously across the nanochannels, and a portion of the Gibbs free energy of mixing can be harvested continuously from the nanochannels by means of the net diffusion current. In the present study, power generation by reverse eletrodialysis in ion-selective nanochannels is numerically investigated by solving the Nernst-Planck equation for the ionic concentrations, the Poisson equation for the electric potential, and the Navier-Stokes equation for the electrolyte velocity simultaneously. We elucidated the effect of various parameters on power generation such as geometry of channel cross section, channel length, hydraulic diameter and the surface charge density etc.

[1]Corresponding Author

F1.00116 An experimental investigation into the effect of Marine Hydrokinetic (MHK) turbine array spacing on turbine efficiency and turbine wake characteristics , NICK STELZENMULLER, ALBERTO ALISEDA, University of Washington — Three 1/45 MHK turbine scale models were tested in a flume at various array spacings. The model turbines were instrumented to measure torque and angular velocity. Incident flow on the turbines and in the wakes was characterized via PIV and ADV measurements. Flow characteristics: mean velocity, turbulence intensity, and vorticity are correlated with turbine performance. Tip speed ratio (TSR) similarity (although not Reynolds number) of the turbines is achieved by controlling the applied load with magnetic brakes inside the model turbine nacelles. Wake characteristics and turbine efficiencies were investigated at a range of TSRs, with the goal of "tuning" an array to maximize overall array efficiency. Grids were placed in the flume upstream of the turbine array in order to change the turbulence intensity of the flow incident to the array. High levels of turbulence intensity in the incident flow is consistent with natural conditions in tidal currents, and has a strong effect on turbine wake dissipation. These experiments used a "reference model" turbine geometry developed for DOE at the National Renewable Energy Laboratory for the purpose of facilitating the comparison of experimental and numerical results in marine hydrokinetic turbine research.

F1.00117 Windblown dust emission, transport and deposition in solar farms , CHUANJIN LAN, ZHEN LI, YANBAO MA, School of Engineering, University of California, Merced — Dust accumulation on solar collectors can significantly reduce the electrical output of solar farms. The presence of solar panel array can significantly accelerate or decelerate wind speed and distort the wind velocity profiles near the ground, which leads to considerable changes in dust emissions, transportation as well as deposition. To examine the effects of solar panels on dust emission, transportation and deposition, the incompressible viscous flow past flat solar panels with ground effect was numerically investigated based on finite volume method. A hybrid approach known as detached-eddy simulation (DES), combining the main features of both large-eddy simulation (LES) and Reynolds-averaged Navier-Stokes (RANS), is utilized to the compute the turbulence flow. Results show how aerolian dust emissions, transport and deposition are affected by wind speeds, solar panel orientation angles and panel geometries.

F1.00118 Detached-eddy simulations of hydrokinetic turbines using actuator disks[1] , DOMENICO SCIOLLA, CRISTIAN ESCAURIAZA, Pontificia Universidad Catolica de Chile — The development of new technologies to harness energy from tidal currents in coastal areas requires an understanding of the interaction of the flow over arbitrary bathymetries and the marine hydrokinetic (MHK) turbines that can be potentially installed at a specific site. When computing realistic flows past multiple MHK devices, numerical models should satisfy the following attributes: (1) Resolve the rich dynamics of the wakes to capture the instantaneous interactions of the turbulent coherent structures; (2) Deal with complex arbitrary bathymetries of the natural channels; and (3) Employ low-cost modeling techniques to incorporate the regions of interest with good resolution and multiple turbine arrangements. In this investigation we simulate the flow past porous disks using the detached-eddy simulation approach (DES). The results show that the model reproduces accurately the mean flow and turbulence statistics of the wakes, and constitutes a powerful tool for analyzing the flow field in realistic conditions using low computational resources. The simulation results are also employed to determine the forces induced by the turbine array on the entire flow, parameterizing its effects to be incorporated in regional-scale models.

[1]Support for this work has been provided by Fondef project D09I1052

F1.00119 Analysis of Tumble and Its Effects on EGR Tolerance for a Gasoline Engine Running at High Loads[1] , JORDAN EASTER, PAULIUS PUZINAUSKAS, TIMOTHY PYLES, University of Alabama — The series hybrid electric vehicle allows for the design of an engine that can run solely at its most efficient point, wide open throttle (WOT). However, at WOT there is an increase in emissions not typically handled in the conventional gasoline engine. Exhaust gas recirculation can be used to reduce emissions if the tolerance of the engine for the exhaust gas is increased. It is hypothesized that tolerance at WOT will increase when there is an increase in in-cylinder turbulence. In this research, aluminum flow guide vanes were inserted in the intake to induce tumble. The flow was examined through the use of PIV techniques and the increase in EGR tolerance was verified with engine testing. PIV images of the flow structure were taken between the intake valves of a modified cylinder designed to mimic bottom dead center. The lift to valve diameters as well as the vane configurations were altered. Engine testing was performed with varying vane configurations, while the EGR percentage was increased until it became difficult to control combustion. It was been found through the engine testing that the flow guide vanes do significantly increase the EGR tolerance as well as combustion stability.

[1]Funding received by the NSF REU Grant 1062611.

F1.00120 In-Depth Absorption of Thermal Radiation in Phase Change Materials[1] , LAURA PERSHERN, St. Cloud State University, MORGAN MINTON, JOHN BAKER, University of Alabama — Performance of a direct absorption solar thermal collector employing a phase change material (PCM) will be directly impacted by the radiant properties of the PCM. Melting and solidification behavior of paraffin, a commonly used PCM, exposed to an external thermal radiation field was investigated. As radiant properties of paraffin are different with respect to solid and liquid phases, the absorption of thermal radiation within the paraffin will be different in the two phases. It is hypothesized that this fact will change the melting and solidification behavior of the paraffin. The solid-liquid interfacial position was recorded as a function of time for heating from above for varying incident radiant intensities. When observed, flow structure was recorded. A one-dimensional heat transfer model was used to gain insight into the temperature behavior with the solid phase.

[1]Funded by the National Science Foundation (Grant No. 1062611)

F1.00121 Flow through porous media with fractal geometry: effect of wettabilty , D. HERNANDEZ, O. CHAVEZ, R. ZENIT, Universidad Nacional Autonoma de Mexico — We experimentally analyze the behavior of two-phase flow through porous media with a fractal geometry. We are interested in this particular case because it has been shown that many oil wells posses a fractal-type porosity structure. In the laboratory, fractal porous media were prepared considering arrays of glass spheres of different diameters and volumetric proportions. To vary the wettability, which is another factor of great importance for oil extraction, the glass spheres was treated with a hydrophobic coating. By measuring the pressure drop and the flow rate, the relative permeability was determined. Additional, some flow visualization experiments were conducted. We found that the relative permeability increases as the wettability decreases: air bubbles tend to remain in contact with the glass surfaces, while the fluid tends avoid them. We also discuss the changes of permeability for fractal and non-fractal porous media.

F1.00122 GEOPHYSICAL –

F1.00123 Simulation based study of the effect of ocean waves on floating wind farm[1] , DI YANG, Civil Engineering, Johns Hopkins University, CHARLES MENEVEAU, Mechanical Engineering & CEAFM, Johns Hopkins University, LIAN SHEN, Civil Engineering, Johns Hopkins University — A hybrid numerical capability is developed for the simulation of floating wind farm offshore, in which large-eddy simulation is performed for wind turbulence, and a potential flow based method is used for the simulation of ocean wavefield. The wind and wave simulations are coupled through a two-way feedback scheme. The effect of wind turbines on the wind field is represented by an actuator disk model. A variety of fully-developed and fetch-limited wind-sea conditions are considered in the study. The simulation results indicate that the offshore wind farm obtains a higher wind power extraction rate under the fully-developed wind-sea condition compared with the fetch-limited condition. This higher extraction rate is caused by the faster propagating waves and the lower sea-surface resistance on the wind when the wind-seas are fully developed. Such wave effect becomes more prominent when the turbine density of the wind farm increases.

[1]DY and LS acknowledge the support of NSF-CBET-1133700. CM acknowledges the support of NSF-CBET-1133800 and NSF-AGS-1045189.

F1.00124 Internal waves and turbulence created by tidal flow over small-scale topographic features , MASOUD JALALI BIDGOLI, NARSIMHA RAPAKA, SUTANU SARKAR, Mechanical and Aerospace Engineering, University of California, San Diego — Numerical simulations are used to study the internal waves and turbulence generated by the oscillation of a barotropic tide over bumps of small length scale where the tidal excursion number, Ex, i.e., the ratio of tidal excursion length to the topographic width is order unity. The objective is to go beyond linear theory to assess the effect of tidal excursion number. Subcritical to supercritical slope angles are considered to investigate if slope angle in conjunction with excursion number leads to different regimes. At low values of Ex, the results agree with Rapaka et al (2011) but there are significant differences in the $Ex = O(1)$ limit. The internal wave field is characterized using spectra, modal analysis and the baroclinic energy budget. Qualitative differences in turbulence generation, phasing and energetics are found when Ex increases.

F1.00125 Effect of the bottom profile on coastal topographic waves[1] , GERARDO RAMÍREZ ROSARIO, LUIS ZAVALA SANSÓN, CICESE Dep.Oceanografia Fisica — We consider linear shallow water equations (LSWE), for a straight coast whose profile is given by $H(x) = \alpha x^s$, where s is a positive real number and x is the distance perpendicular to the coast. We show how the LSWE can be transformed in to an ordinary differential equation, which is solved by perturbation methods. The perturbation term depends on wave frequency, the Coriolis parameter and geometric features of the coast and decays exponentially with offshore distance. The solutions of the unperturbed problem are the associated Laguerre polynomials. These polynomials are the basis for finding approximate solutions of the perturbed problem. For the case $s = 1$ the method recovers the solution reported in the literature. The dispersion relation of the unperturbed problem corresponds to that obtained with the rigid lid approximation. The dispersion relation shows that for small s sub-inertial modes are less affected by topography than super-inertial modes. However, for large s sub-inertial modes are more affected than super-inertial. An interesting case is $s = 2$, since the eigen frequencies do not depend on wave number.

[1]This work is supported by the National Council of Science and Technology (CONACyT, México) under PhD. scholarship 224928.

F1.00126 Numerical experiments of atmospheric flow off Baja California, México , CARLOS TORRES[1], SERGIO LARIOS[2], ADAN MEJIA, JAIME GARCIA, EDUARDO GIL, Instituto de Investigaciones Oceanológicas, UABC — In order to simulate flow structures suggested by satellite images of cloud trails off Baja California, the momentum primitive equations describing an atmospheric flow over that region are solved numerically on a boundary-fitted grid. Numerical experiments are conducted for several flow conditions. Results show a remarkable agreement to available observations.

[1]also at Mexico National Oceanographic Data Center, UABC
[2]also at Mexico National Oceanographic Data Center, UABC

F1.00127 Thermally Driven Flow in a Mock Street Canyon , ANN DALLMAN, University of Notre Dame, SIGURDUR MAGNUSSON, LESLIE NORFORD, MIT, HARINDRA J.S. FERNANDO, University of Notre Dame, DARA ENTEKHABI, REX BRITTER, MIT, SHANSHAN PAN, Nanyang Techinal University — Under conditions of low synoptic winds and high solar radiation, non-uniform heating of building walls and the ground in an urban street canyon induces thermally-driven airflow. These effects have mainly been studied using wind-tunnel experiments and numerical models, but only a few field-scale experiments have been performed. However, this is an important topic of interest because of its implications for air quality and emergency response planning. A field experiment was carried out in collaboration between the Singapore-MIT Alliance for Research and Technology (SMART) and the University of Notre Dame. The study was conducted on the campus of Nanyang Technical University in Singapore, and consisted of an 'idealized' building canyon constructed with two rows of shipping containers aligned in the North-South direction. The site was carefully instrumented with sonic anemometers (for wind speed and direction and virtual temperature), weather stations (wind speed and direction, temperature, relative humidity, pressure, and rain fall), and thermocouples (surface temperature of buildings). Measurements were recorded for 9 days, which included periods of sunshine and high convective activity that created thermal circulation between the buildings. Using a fog machine, flow visualization was carried out to observe circulation patterns. An overview of the experiment and the results will be presented.

F1.00128 Ventilation Exhaust Power Recovery Design , JEREMY YANDELL, ALBERTO ALISEDA, University of Washington — Due to the expense of designing ductwork and exhaust fans to meet the exact desired flow rate for building exhaust, there is wasted energy that is unrecovered when exhausted to the atmosphere. By designing a small diameter wind turbine the kinetic energy in the exhaust stream can be recovered and power provided back into the building. Unlike large scale commercial wind turbines that must be designed to provide power from a large range of wind speeds and directions, this smaller scale turbine can be optimized for a single constant wind speed with no variation in direction. The critical component is to prevent backpressure feeding through the system and increasing the load on the exhaust fan. This design project began with the theoretical airfoil and blade design, followed by modeling the system in fluid dynamics software, a full CAD design was created and modified for the selected manufacturing process, prototype creation and testing will be completed both in a wind tunnel and in a real environment, and the completed data will be compared with theoretical and computational results. Note: There is a patent pending for this design and concept.

F1.00129 Computational Analysis of Flow Field Hydrodynamics Between Identical Spheres in Water , GAURANG SHYAM LIMACHIA, None — Stokes' Law is the foundation of modern hydrodynamics. The introduction of rigid particles into a uniform flow makes the governing equations highly complex; consequently, these equations are nearly impossible to resolve analytically. As a result, solutions are often found for specific instances through numerical analysis. This paper performs such numerical analysis for rigid spheres suspended in a single fluid with uniform upward flow. The perturbation of the velocity field was found to be greatest when the spheres were within 1.30 Diameters of each other. In contrast, the spheres moved independently with minimum interaction at distances greater than 5 Diameters apart. This research helps us identify ideal interaction distances for suspended objects and understand the natural orientation of real organisms traveling in real fluids in the real world.

Sunday, November 18, 2012 7:00PM - 8:30PM –

Session F2 Welcome Reception Sails Pavilion

7:00PM F2.00001 WELCOME RECEPTION –

Monday, November 19, 2012 8:00AM - 10:10AM –

Session G1 Geophysical: Atmospheric II 22 - Chair: Alberto Aliseda, University of Washington

8:00AM G1.00001 Rain-induced dissipation in hurricanes[1] , TAPAN SABUWALA, GUSTAVO GIOIA, PINAKI CHAKRABORTY, Okinawa Institute of Science and Technology — Hurricanes originate from a potent mix of atmospheric and oceanic conditions, and manifest intensely swirling winds and torrential rains. The drag forces on the falling raindrops act to dissipate energy, which, in the context of global precipitation, has been shown to play a key role in global atmospheric circulation. And yet, the role of rain-induced dissipation in the energetics of a hurricane remains uncharted. Here, using dimensional analysis and satellite data assimilated from the Tropical Rainfall Measuring Mission, we propose a simple model of rain-induced dissipation in a hurricane. We modify Emanuel's idealized heat engine model of hurricanes by incorporating the rain-induced dissipation and predict the maximum intensity a hurricane can achieve for a given set of atmospheric and oceanic conditions. We find that the modified model predictions are closer to the observed data as compared with Emanuel's model. Further, we use the modified model to predict inter-annual trends in various metrics of hurricane activity in the North Atlantic basin and show that the model predictions compare well with the observed trends. We conclude that rain-induced plays a significant role in the energetics of hurricanes and should be incorporated in global climate models.

[1]This research was partially supported by funds from the Okinawa Institute of Science and Technology

8:13AM G1.00002 Impact of the asymmetric dynamical processes of the structure and intensity change of two-dimensional hurricane-like vortices , KONSTANTINOS MENELAOU, M.K. YAU, McGill University, YOSVANY MARTINEZ, Researcher — In this study, a simple two-dimensional (2D) unforced barotropic model is used to study the asymmetric dynamics of the hurricane-inner core region and assess their impact on the structure and intensity change. Two sets of experiments are conducted starting with a ring of enhanced vorticity to mimic intense mature hurricane-like vortices, that is perturbed by an external impulse. The theory of empirical normal modes (ENM), and the Eliassen-Palm (EP) flux theorem is then applied to extract the dominant wave modes from the dataset and diagnose their kinematics, structure, and impact on the hurricane-like vortex. From the first experiment, it is found that the evolution and the lifetime of an elliptical eyewall may be controlled by the inviscid damping of sheared vortex Rossby waves (VRWs) or quasimodes. The critical radius and the structure of the quasimode obtained by the ENM analysis is shown to be consistent with the predictions of a linear eigenmode analysis of small perturbations. From the second experiment, it is found that the outward propagating VRWs that arise due to barotropic instability and the inward mixing of high vorticity, organizes into secondary ring of enhance vorticity that contains a secondary wind maximum. Sensitivity tests performed on the spatial extent of the initial external impulse verifies the robustness of the results. The fact that the secondary eyewall occurs close to the critical radius of some of the dominant modes emphasize the important role played by the VRWs.

8:26AM G1.00003 Radiatively Driven Turbulence at the Cloud Top , ALBERTO DE LOZAR, JUAN PEDRO MELLADO, Max Planck Institute for Meteorology — We use Direct Numerical Simulations to investigate a radiatively-driven smoke cloud-top mixing layer. This configuration mimics relevant aspects of stratocumuls clouds, in particular the mixing across an inversion that bounds a radiatively driven turbulent flow. A 1D formulation is employed for the radiation calculations. Below the inversion a convective boundary layer propagates downwards into the cloud-bulk. The convective boundary layer decouples from the inversion properties other than the injected buoyancy flux. This buoyancy flux is equal to the total radiative cooling minus the cooling of the inversion layer where the cloud mixes with the free atmosphere. An exact equation at a properly defined inversion point divides the inversion cooling into three components: a molecular flux, a turbulent flux and the direct radiative cooling by the smoke inside the inversion layer. The normalized turbulent flux levels to a constant value (0.175 ± 0.05), which is independent of the stratification. As suggested by earlies studies, we observe that the turbulent entrainment only occurs at the small scales and that eddies larger than four optical lengths ($50\,\mathrm{m}$ in a typical DYCOMS-II cloud) perform little or no entrainment.

8:39AM G1.00004 Large-eddy simulations of contrail-to-cirrus transition in atmospheric turbulence , ROBERTO PAOLI, ODILE THOURON, JORIS PICOT, 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 cirrus clouds, they can persist for hours and become almost indistinguishable from natural cirrus. This talk focuses on the role of atmospheric turbulence in determining the characteristics of these "contrail cirrus." Large-eddy simulations are carried out using the atmospheric model Meso-NH with the goal of identifying the processes driving the contrail-to-cirrus transition as a function of contrail age. To that end, the effects of atmospheric turbulence, microphysics, and radiative transfer are analyzed separately. Turbulent fields are first generated by means of a stochastic forcing technique that reproduces the atmospheric conditions encountered in the upper troposphere. Contrails generated by a model aircraft are then inserted on the top of these fields. Finally, ice microphysics and radiative transfer are activated to find out on which spatial and temporal scales the vertical motion prevails over the essentially horizontal motion induced by atmospheric turbulent diffusion.

8:52AM G1.00005 On the evolution of stratified turbulent clouds , ANDREA MAFFIOLI, PETER DAVIDSON, Department of Engineering, University of Cambridge, STUART DALZIEL, DAMTP, University of Cambridge, NEDUNCHEZHIAN SWAMINATHAN, Department of Engineering, University of Cambridge — The evolution of a turbulent cloud in a stratified fluid is studied by means of direct numerical simulations. The focus of the study is on the edge dynamics occurring between the turbulence and the quiescent region surrounding it. By comparing isosurface plots of the materially conserved potential vorticity Π and the u_x velocity component (x is horizontal) it is possible to divide the edge flow into fluid intrusions and horizontally-travelling wave-packets. The 3D structure of the intrusions and the wave-packets is similar, both structures being pancake-like, and the only difference is in their extension away from the turbulent cloud. Individual wave-packets were tracked and it was found that their group speed agrees with the theoretical group speed relation for linear internal gravity waves. The wave-packets can therefore be thought of as quasi-linear finite-amplitude internal gravity waves. The kinetic energy radiated away by the waves was measured and it was found to be 15–20% of the total kinetic energy in the numerical domain. As a result, horizontally-travelling waves set off during a localized turbulence episode in the atmosphere could be important in the context of meteorology as they alter the energy and momentum budgets in different regions of the atmosphere.

9:05AM G1.00006 Effect of inertia of water droplets on a turbulent cloud: a toy model , RAMA GOVINDARAJAN, RAVICHANDRAN SIVARAMAKRISHNAN, TIFR Centre for Interdisciplinary Sciences, Tata Institute of Fundamental Research, Hyderabad — We ask whether the fact that water droplets are inertial can affect the upward trajectory of a cloud. We answer in the affirmative using a toy model, where the turbulent cloud is represented by a distribution of point vortices. Viscosity is neglected and the water droplets are assumed to be point particles whose inertia only gives rise to a drag against the flow. The growing water droplets are shown to cluster in a curtain-like structure along the sides of the cloud as it rises through the atmosphere, causing repeated cycles of nucleation, growth and departure of water droplets in the central region of the cloud. The net effect is a slowing down of the loss of water vapour, and the resulting "slow release" of buoyancy allows the cloud to attain a higher height.

9:18AM G1.00007 Preferential accumulation, enhanced relative velocity and gravitational settling due to inertial droplet interactions with turbulence , COLIN BATESON, ALBERTO ALISEDA, University of Washington — We are exploring the hypothesis that, during warm-rain formation, turbulence-induced collisions can explain the size gap between the limit of condensational growth and the onset of gravitational collisions and sedimentation. We use wind tunnel experiments to study the evolution of water droplets in homogeneous, isotropic, slowly decaying grid turbulence. We analyze the preferential concentration and the enhanced relative velocity of droplets in the $10 - 200\ \mu m$ range due to their inertial interactions with the underlying turbulence. Data from Phase Doppler Particle Analysis and flow visualizations provide insight into the droplet relative velocity and settling velocity fields. We focus on those fields' dependency on droplet Stokes number and local concentration. Recent improvements to our experimental setup allow for high-magnification, high-speed imaging of the flow inside the wind tunnel. We use these images to observe near-droplet dynamics and collision events, with the ultimate goal of formulating a model for droplet collision-coalescence efficiency that can be used in numerical simulations and parameterizations of turbulence-induced droplet collisions.

9:31AM G1.00008 E-epsilon turbulence closure for sea spray-laden marine atmospheric boundary layer in high wind conditions[1] , YEVGENII RASTIGEJEV, North Carolina A&T State University, SERGEY SUSLOV, Swinburne University of Technology — In-depth understanding and accurate modeling of the interaction between sea spray and a turbulent airflow under high-wind conditions is essential for improving intensity forecasts of hurricanes and severe storms. Here we consider the E-epsilon turbulence closure for the spray-stratified atmospheric marine boundary layer. Our mathematical model accounts for turbulent kinetic energy transport in the vertical direction, the dependence of the turbulent mixing length on the spray stratification and the spray inertia. It is shown that accounting for all these physical factors is important since none of them dominate for all possible hurricane conditions. The obtained analytical and numerical solutions show significant differences between the current E-epsilon model and the lower order Turbulent Kinetic Energy (TKE) and Monin-Obukhov (MO) similarity models considered previously. It is demonstrated that the air turbulence suppression by the spray causes an acceleration of the airflow and a reduction of air-sea drag coefficient that is qualitatively consistent with resent experimental observations.

[1]The work of YR was supported by NSF, HRD-1036563 grant.

9:44AM G1.00009 Wave-induced mean flow at an interface , JOHN MCHUGH, University of New Hampshire, ROBERT SHARMAN, National Center for Atmospheric Research — A vertical packet of internal waves that are horizontally periodic will generate a mean flow that has the same sense as the group velocity of the incident waves. When the wave packet impinges on a density-gradient interface the waves are partially reflected and the wave-induced mean flow is enhanced just under the interface. A density-gradient interface has continuous density but discontinuous buoyancy frequency, and is an idealization of Earth's tropopause. Here we consider waves generated by flow past an isolated object, and maintain a vertical packet by introducing the obstacle gradually. The resulting waves are confined horizontally over a narrow interval and hence are not slowly varying in the horizontal. Nonlinear simulations show that the mean flow at the interface has a component with the same sense as the wave group velocity above the mean position of the interface, but also a component with the opposite sense just below the mean interfacial position. This combination establishes a wave-induced circulation at the interface.

9:57AM G1.00010 Modes in rotating troposphere with a leak , LYUBOV CHUMAKOVA, RODOLFO ROSALES, MIT, ESTEBAN TABAK, CIMS, NYU — We study how the rigid lid modes change, if the effect of waves leaking into the stratosphere from the troposphere is taken into account in the linear, hydrostatic, incompressible, rotating case. We present an effective boundary condition which accounts for the leak and the resulting "leaky modes." This work is partially supported by NSF 0903008 (LC), DMS 1007967 and DMS 0907955 (RR), and DMS 0908077 (ET).

Monday, November 19, 2012 8:00AM - 10:10AM –

Session G2 Convection and Buoyancy-Driven Flows IV 23A - Chair: Herman Clercx, Technical University Eindhoven

8:00AM G2.00001 Horizontal convection with mechanical stirring , ROSS GRIFFITHS, Australian National University, KIAL STEWART, Johns Hopkins University, GRAHAM HUGHES, Australian National University — The effects of turbulent mixing on convective circulation forced by a horizontal gradient of buoyancy at the surface is examined using laboratory experiments in which a salt flux is introduced at the surface, at one end of a box, and a freshwater buoyancy condition is applied over the rest of the surface. Horizontal rods are oscillated and yo-yoed continuously through the water column, providing a diffusivity that can be calibrated. The convection reaches a stationary state having zero net salt flux. We find that for small stirring rates the small but finite volume flux from the dense source is significant and a virtual source correction is required to take this into account. The density stratification and overturning volume transport are consistent with a theoretical model for high Rayleigh numbers: the transport ψ increases with diffusivity κ ($\psi \mathrm{g} \sim \mathrm{g}\kappa^{1/4}$). The results show that vertical mixing in the boundary layer is important, particularly in setting the density of the interior and the overturning rate. However, interior mixing is unimportant, which raises an interesting question over whether abyssal mixing rates in the ocean play any significant role in setting the abyssal ocean density or the transport in the Meridional Overturning Circulation.

8:13AM G2.00002 The Effects of Surface Stress on Horizontal Convection[1] , KATARZYNA E. MATUSIK, STEFAN G. LLEWELLYN SMITH, University of California, San Diego — Laboratory experiments have been designed to investigate the effects of a surface stress on horizontal convection. In the ocean, a zonal wind stress drives a meridional Ekman flow due to the effects of rotation. We explore features of horizontal convection in the presence of a surface stress that is imposed in opposition to the buoyancy-driven circulation. The buoyancy flux is achieved by injecting a plume of dense water into a fresh-water tank, while continuously maintaining a fresh-water surface boundary condition. The magnitude of the stress is varied by adjusting the flow rate of fresh water traversing along the surface; this stress is run in parallel to the north-south buoyancy gradient in order to simulate a 2D non-rotating system. We measure the steady-state density field using the synthetic schlieren technique, and the vertical and horizontal velocities are determined by PIV techniques. The boundary layer is resolved with conductivity probe measurements. The addition of a surface stress to horizontal convection may offer insight into the effects of wind on the ocean surface, namely the implications of a kinetic energy source on the overall energetics of the circulation.

[1]NSF OCE-0926481

8:26AM G2.00003 Advection and buoyancy-induced turbulent diffusion in a narrow vertical tank , DAAN D.J.A. VAN SOMMEREN, C.P. CAULFIELD, University of Cambridge, BP Institute and DAMTP, ANDREW W. WOODS, University of Cambridge, BP Institute — We describe experiments to examine the turbulent mixing due to a source with constant buoyancy flux B_s at the top of a vertical tank (with dimensions $40d \times d \times d$) in which an upward flow with speed u_b is present. Dense source fluid vigorously mixes with the less dense fluid of the upward flow. The mixed region of fluid is characterised by an unstable density gradient, which drives a turbulent flow which is dominated by eddies of the size of the width of the tank. These turbulent eddies are associated with the downward flux of dense fluid, which is modelled as a diffusive process. The upward flow with speed u_b is associated with the advective upward flux of dense fluid. During the late-time steady mixing phase, the diffusive and advective flux of dense fluid are in balance. The mixed region then extends a distance $h_{st} = 3d\lambda^{4/3}/\mathrm{Frn}$ from the top of the tank, where Fr is a Froude number defined by $\mathrm{Fr} = \mathrm{u_b}/(\mathrm{B_s}^{1/3}\mathrm{d}^{-1/3})$, and λ is an $O(1)$ constant relating the width of the tank to the characteristic mixing length of the turbulent eddies. We use a dye-attenuation technique to obtain vertical profiles of the horizontally-averaged reduced gravity, and show a good agreement between experiments and theory.

8:39AM G2.00004 Simulation of highly-unsteady hydrothermal convection above the critical temperature in the deep sea[1] , SATOKO KOMURASAKI, College of Science and Technology, Nihon University — Eruption of geothermally heated water from the hydrothermal vent in deep oceans of depth over 2,000 meters is numerically simulated. The hydrostatic pressure of water is assumed to be over 200 atmospheres, and temperature of heated water occasionally more than 300°C. Under these conditions, a part of heated water can be in the supercritical state, and the physical properties can change significantly by the temperature. The compressible Navier-Stokes equations are solved using a method for the incompressible equations under the assumption that the pressure is almost constant at the hydrostatic pressure and the density is a function of the temperature. The equations are approximated by the multidirectional finite difference method, and for the highly-unsteady-flow computation, KK scheme and a hybrid upwind scheme are used to stabilize the high-accuracy computation. Computational results show that complexity and the unsteadiness of the flow are significantly influenced by whether the issuing high temperature water is in the supercritical state or not.

[1]This work was partially supported by Grant-in-Aid for Scientific Research from MEXT/JSPS (22740261).

8:52AM G2.00005 A Computational Investigation of Mixed Convection in Microscale Flows[1] , RUSTEM BILYALOV, JOHN BAKER, University of Alabama — In order to study mixed convective heat transfer associated with flow in a vertical microscale channel, a two-dimensional computational model was used. A temperature difference was established in the direction normal to the flow by assuming that each of the channel walls is at a constant, but different temperature. The microscale geometry resulted in finite Knudsen number flows in the so-called slip flow regime. The Maxwell velocity-slip and temperature-jump boundary conditions were applied at the channel walls. The flow structure was visualized using contour plots of temperature and pressure as well as velocity vector plots. Flows corresponding to Knudsen numbers in the range of 0.01 through 0.1 and an Archimedes number in the range of 0.1 to 10 were considered. Mixed convective heat transfer, for both assisting and opposing conditions, was characterized using the Nusselt number.

[1]Funding recieved by NSF REU Grant 1062611 and the von Braun Center for Science and Innovation.

9:05AM G2.00006 Buoyancy-driven instability of a miscible horizontal displacement in a Hele-Shaw cell , F. HAUDIN, L.A. RIOLFO, Nonlinear Physical Chemistry Unit, Universite Libre de Bruxelles (ULB), Brussels, Belgium, B. KNAEPEN, Statistical and Plasma Physics group, Universite Libre de Bruxelles (ULB), Brussels, Belgium, A. DE WIT, Nonlinear Physical Chemistry Unit, Universite Libre de Bruxelles (ULB), Brussels, Belgium — In Hele-Shaw cells, viscous fingers are forming when a fluid is injected into a more viscous one. If the two fluids are reversed, with the less mobile fluid injected into the low viscosity one, the situation is expected to be stable from a viscous point of view. Nevertheless, a destabilization of the interface can be observed due to a buoyancy-driven effect if a density difference exists between the two miscible fluids. As a result, the Poiseuille profile established in the gap of the cell locally destabilizes and convection rolls are forming. In a view from above, a striped pattern is observed at the miscible interface between the two fluids. To characterize the development of this instability, we have performed an experimental study of viscously stable miscible displacements in a Hele-Shaw cell with radial injection. The displacing fluids are aqueous solutions of glycerol and the displaced ones are either dyed water or dyed glycerol solutions. The way the relative properties of the two fluids is influencing the onset time of the instability and the characteristic size of the pattern is studied. The influence of the gap width and of the flow rate on the buoyantly unstable dynamics is also characterized.

9:18AM G2.00007 Thermal convection in a nonlinear non-Newtonian magnetic fluid , HARALD PLEINER, Max Planck Institute for Polymer Research, Mainz, Germany, DAVID LAROZE, Max Planck Institute for Polymer Research, Mainz, Germany and Instituto de Alta Investigacion, Universidad de Tarapaca, Arica, Chile — We report theoretical and numerical results on the convection of a magnetic fluid in a viscoelastic carrier liquid. The non-Newtonian material properties are taken care of by a general hydrodynamic nonlinear viscoelastic model [1] that contains, but is more general than the standard Oldroyd and Giesekus phenomenological rheological equation for the stress tensor. We calculate the linear threshold for both idealized and rigid boundary conditions and make the comparison with the linear Oldroyd magnetic fluid [2]. In order to explore the nonlinear behavior we perform a truncated Galerkin expansion obtaining a generalized Lorenz system. We find numerically the system's stationary, periodic and chaotic regimes by investigating power spectra and Lyapunov exponents. Finally, we give a phase diagram depicting the various types of dynamical behavior as a function of the Rayleigh number and the viscoelastic material parameters.

[1] H. Pleiner, M. Liu, H.R. Brand, Rheol. Acta. 43, 502 (2004).
[2] L.M. Pérez, J. Bragard, D. Laroze, J. Martinez-Mardones, H. Pleiner, J. Mag. Mag. Mat. 323, 691 (2011).

9:31AM G2.00008 Simulation of the flow and heat exchange in a cylindrical solar chemical reactor , MANUEL RAMÍREZ-CABRERA, EDUARDO RAMOS, CIE-UNAM — In this work, we present the simulation of the flow inside a cylindrical container filled with an optically participating medium. The motion is generated by the combined effect of the forced convection due to an axial pressure gradient and a natural convective flow induced by a beam of heat radiation that enters into the container though a transparent window located on one of the plane surfaces. The bouyancy force is considered perpendicular to the cylinder axis. The simulation is based on the simultaneous solution of the mass, momentum and energy conservation equations, coupled with the radiation intensity transfer equation. The flow patterns and temperature distributions as functions of the pressure gradient are described to identify the parameters required to maximize the heat absorbed and to obtain a specific temperature field for potential applications. The physical conditions considered are similar to those found in a cylindrical Solar-driven water-splitting thermochemical reactor and it is expected that the results will be useful to determine optimum design parameters.

9:44AM G2.00009 New type of thermal waves in a vertical layer of magneto-polarizable nano-suspension: theory and experiment , SERGEY A. SUSLOV, Swinburne University of Technology, ALEXANDRA A. BOZHKO, GENNADY F. PUTIN, ALEXANDER S. SIDOROV, Perm State National Research University — Study of Boussinesq convection in a vertical differentially heated fluid layer is one of classical problems in hydrodynamics. It is well known that as the value of fluid's Grashof number increases the basic flow velocity profile becomes unstable with respect to stationary shear-driven disturbances (at Prandtl numbers $Pr < 12.5$) or thermogravitational waves propagating vertically (at larger values of Prandtl number). However linear stability studies of a similar flow of magnetopolarizable nanosuspensions (ferrofluids) placed in a uniform magnetic field perpendicular to a fluid layer predicted the existence of a new type of instability, oblique waves, that arise due to the differential local magnetisation of a non-uniformly heated fluid. The existence of such (thermomagnetic) waves has now been confirmed experimentally using a kerosene-based ferrofluid with magnetite particles of the average size of 10 nm stabilized with oleic acid. The heat transfer rate measurements using thermocouples and flow visualization using a thermosensitive film and an infrared camera have been performed. Perturbation energy analysis has been used to determine the physical nature of various observed instability patterns and quantitatively distinguish between thermogravitational and thermomagnetic waves.

9:57AM G2.00010 DEP thermal convection in annular geometry under microgravity conditions[1] , HARUNORI YOSHIKAWA, OLIVIER CRUMEYROLLE, INNOCENT MUTABAZI, Laboratoire Ondes et Milieux Complexes, UMR 6294 CNRS-Universite du Havre — Thermal convection driven by the dielectrophoretic force is investigated in annular geometry in microgravity environments. A radial heating and a radial alternating electric field are imposed on a dielectric fluid layer filling the gap of two concentric infinite-length cylinders. The resulting dielectric force field is regarded as spatially varying radial gravity that can develop thermal convection. The linear stability problem of a purely conductive basic state is solved by a spectral-collocation method for both axisymmetric and non-axisymmetric disturbances. A stationary non-axisymmetric mode becomes first unstable at a critical Rayleigh number to develop convection. The stability boundary shows asymmetry with respect to heating direction. For an outward heating the critical value approaches that of the Rayleigh-Bénard problem (1708) as the gap size decreases, while it converges to larger values in the narrow gap limit. For an inward heating the instability occurs only when the gap is narrower than a certain value. The critical number diverges with increasing the gap size. Instability mechanism is examined from energetic viewpoints. The feedback of electric field to temperature disturbances is found to stabilize the conductive state for narrow gaps.

[1]This work has been partly supported by the CNES, the CNRS and the FEDER.

Monday, November 19, 2012 8:00AM - 10:10AM –

Session G3 Multiphase Flows: Numerical Methods I 23B - Chair: Carlos Coimbra, University of California, San Diego

8:00AM G3.00001 Effect of gravity on particle dispersion in upward gas turbulent channel flow[1]

, YOICHI MITO, Kitami Institute of Technology — Fully-developed concentration fields of solid particles with a large range of inertial time constants in a plane channel in which gas is flowing turbulently in the upward direction are calculated by using direct numerical simulation to calculate the gas velocities seen by particles and a Lagrangian method to calculate trajectories of particles. The objective is to examine the effects of gravity and of inertia, both of which represent trajectory mechanisms that decrease turbulent dispersion, on particle transport in wall-bounded turbulent flow. The frictional Reynolds number is 150. Density ratio is 1000. The Stokesian inertial time constants made dimensionless with the friction velocity and kinematic viscosity are 5, 10, 20, 40, 100, 200. Three Froude numbers, Fr = 0.02, 5 and infinity, are considered. Forces exerted by particles on the gas and inter-particle collisions are not considered. Effect of gravity on particle dispersion is not seen at Fr $\geq$ 5. The particle turbulence decreases due to the effect of gravity at Fr = 0.02. The effect of gravity increases with increasing particle inertia and with increasing the distance from the wall. It disappears in the viscous wall region where particles are disengaged from gas turbulence structures due to their inertia.

[1]This work is supported by JSPS KAKENHI 22560154.

8:13AM G3.00002 ABSTRACT WITHDRAWN –

8:26AM G3.00003 Dynamics of a cylinder plunging into liquid: a numerical study[1]

, HANG DING, University of Science & Technology of China — The impact of a cylinder on a liquid surface and subsequent events are investigated numerically. The flows are resolved by solving the Navier-Stokes equations and the Cahn-Hilliard equation. Moving contact lines are modeled by a diffuse interface model (Seppecher 1996; Jaqcmin 2000), and contact-angle hysteresis is included (Ding&Spelt 2008). The method is validated by comparison to the experiments by Aristoff and Bush (2009). Our studies focus on the dynamics of the waves induced by the impact and the cavity collapse behind the cylinder. A variety of parameters affect the flow behaviors such as wettability, impact speed, viscosity etc. Their effects on the transition of the flow phenomena are investigated through parametric simulations over relevant ranges of Weber and Reynolds numbers and contact angles.

[1]This work is supposed by the 100 Talents Program of the Chinese Academy of Sciences and the National Natural Science Foundation of China (Grant No. 11172294).

8:39AM G3.00004 A conservative volume of fluid method for general grids in three dimensions[1]

, CHRISTOPHER IVEY, PARVIZ MOIN, Center for Turbulence Research, Stanford University — 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. The advection scheme is formally first order in volume fraction due to is upwinding nature; however, kinematic test cases performed on grids of varying structure and density demonstrate that the accuracy is between first and second order and nominally compares well with contemporary algorithms.

[1]Supported by the DOE CSGF (grant number DE-FG02-97ER25308).

8:52AM G3.00005 Improved connectivity free front tracking method for modeling contact lines in multiphase flow

, CHU WANG, LUCY ZHANG, Rensselaer Polytechnic Institute — A numerical algorithm is developed to simulate multiphase flows with contact lines by coupling dynamic contact line model into the connectivity free front tracking method (CFFT). The contact angle is specified dynamically with an empirical correlation related to the contact line velocity. The hysteresis of the contact angles is also included in the algorithm. This contact line model is coupled to the CFFT conveniently for its explicit representation of the interface. Also, the CFFT does not require the connectivity of the interfacial points to construct the indicator field by ensuring a constant indicator, e.g. I=0.5 coinciding with interface. The topology change of the interface including the contact lines can be treated automatically by adopting a simple points-regeneration scheme. RKPM interpolations are used to achieve better accuracy when constructing the indicator and calculating the normal for contact lines where the interfacial points are connected to the solid wall. Several test cases are performed to validate the method to show its accuracy and capability to simulate multiphase flows with contact lines that undergo frequent topology changes.

9:05AM G3.00006 Shock-driven formation of a cloud from particles swept off a surface[1]

, PATRICK WAYNE, TENNILLE BERNARD, CLINT CORBIN, GARRETT KUEHNER, PETER VOROBIEFF, C. RANDALL TRUMAN, The University of New Mexico, HUGH SMYTH, ANDY MALONEY, University of Texas - Austin — We present an experimental study of respirable particle advection in shock-driven flow. Particles of specific size ($\leq$ 5 μm) were ultrasonically deposited on surface samples, with sample roughness and other characteristics well-known. Then the samples were exposed to normal shocks at Mach number $\sim$1.67. Time-resolved visualizations of the resulting particle clouds provide insights into the physics of the flow. As the clouds evolve, they apparently extend into the flow beyond the wall boundary layer. Several interesting features have been observed, including formation of shear-driven Kelvin-Helmholtz instability on the edge of the cloud. Initial observations suggest a prominent relationship between the force of adhesion between the particles and the surface on one hand and the propagation speed of the particle cloud on the other hand.

[1]This research is supported by DTRA.

9:18AM G3.00007 A multiphase flow solver with adaptive mesh refinement[1]

, KEEGAN DELANEY, ELIAS BALARAS, The George Washington University, ZHIPENG QIN, AMIR RIAZ, University of Maryland — We will present a scalable Navier-Stokes solver applicable to multiphase incompressible flows. The solver employs Level Set techniques to sharply define the interface between different phases (i.e. air and water). 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. Scalable linear solvers are used to solve the variable coefficient Poisson pressure equation on large core counts. 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. The solver takes advantage of Adaptive Mesh Refinement (AMR) to reduce overall cell count in the solution domain, thus reducing computational time. This feature allows for sufficient resolution of complex interfacial features without over-resolving areas of no interest. In the present work, the mesh is selectively refined around the multiphase interface, which is evolving in time. A wide range of multiphase problems will be presented to demonstrate the accuracy and efficiency of the solver.

[1]Work supported by ONR

9:31AM G3.00008 A lattice based approach for simulation of multiphase flows with phase transitions[1] , ABDELAZIZ ALIAT, PRAKASH VEDULA, University of Oklahoma, Norman — We present a lattice based approach to address challenges due to nonequilibrium behavior in liquid-vapor flows with phase transitions. The effects of phase transitions are accounted for via an interaction potential that is treated as a combination of a short range repulsion model based on hard sphere interactions and a long range attraction tail based on mean field models. Particle distribution functions are evolved based on the Boltzmann equation with the full collision operator and a self-consistent force field. Our numerical implementation of this approach involves quadrature-based analytical approximations of moments due to the full collision operator and second-order accurate approximations to convective fluxes (including flux limiters). The results obtained from this approach will be compared with those from other approaches based on standard Lattice Boltzmann Method for simulation of phase transitions in selected canonical flows using different mean-field models. Generalizations of our proposed approach for accurate simulation of heat transfer rates based on high-order lattice representations will also be discussed.

[1]This work is supported by AFRL and UES Inc.

9:44AM G3.00009 Behaviour of a binary circulating liquid-fluidized bed: experiments and CFD modeling , MARCO ROTONDI, Dipartimento di Ingegneria Civile, Chimica e Ambientale, Universitàdegli Studi di Genova, LUCA MAZZEI, PAOLA LETTIERI, Department of Chemical Engineering, University College of London, RENZO DI FELICE, Dipartimento di Ingegneria Civile, Chimica e Ambientale, Universitàdegli Studi di Genova — The literature containing experimental data about binary solid suspensions homogeneously dispersed into liquids is scarce. Having analysed these systems experimentally, we present our findings to investigate the fluid-particle and particle-particle interaction forces. The experimental apparatus is a closed loop where tap water and two different kinds of solids circulate thanks to the action of a centrifugal pump. The fluid moves through PVC tube from the pump to a Plexiglas column of 140 mm internal diameter. Another PVC tube connects the top of the column to the pump, allowing water recirculation. We charged two kinds of solid particles, different in diameter and/or density, inside the system. Two fluidized beds are distinguished inside the column when the velocity is low. With a proper choice of solids, when the velocity increases, one circulates, while the other remains in a fluidized steady state inside the column. We tested different couples of solids in different quantities over a range of fluid velocities, finding new experimental data for binary-solid liquid suspensions. We also simulated the behaviour of the physical system using a multifluid Eulerian computational fluid dynamics (CFD) commercial tool, investigating how mesh size, turbulent flow and periodic boundary influence the simulation results.

9:57AM G3.00010 ABSTRACT WITHDRAWN –

Monday, November 19, 2012 8:00AM - 10:10AM –

Session G4 Drops IV 23C - Chair: Sigurdur Thoroddsen, King Abdullah University of Science and Technology, Saudi Arabia

8:00AM G4.00001 The breakup of thin air films caught under impacting drops , SIGURDUR THORODDSEN, MARIE-JEAN THORAVAL, King Abdullah University of Science and Technology, Saudi Arabia, KOHSEI TAKEHARA, T. GOJI ETOH, Kinki University, Japan — When a drop impacts a pool at very low velocities V, an air layer cushions the impact and prevents immediate contact. This air layer is stretched into a hemispheric shape and thins to a submicron thickness. We use silicone oils, where these films are more stable than for water [Saylor & Bounds (2012), *AIChE J.*, online: *doi 10.1002/aic.13764*]. We observe three main breakup mechanisms which are imprinted onto the micro-bubble morphology. First, for lowest V the film ruptures at isolated holes which grow rapidly, leaving *bubble necklaces* where their edges meet. Based on micro-bubble volumes, we show the film breaks by van der Waals, when its thickness $\sim$ 100 nm. Secondly, for slightly larger V a ring of holes appearing a fixed depth, where the film is thinnest, producing *bubble chandeliers*. Finally, for larger V an air jet within the drop, ruptures it at the bottom tip, in an axisymmetric breakup. We measure the rupture speed and find that for very viscous liquids, the breakup moves faster than the capillary-viscous velocity, through the repeated ruptures. [Thoroddsen, Thoraval, Takehara & Etoh (2012), *J. Fluid Mech.* online: *doi:10.1017/jfm.2012.319*].

8:13AM G4.00002 Toroidal bubble entrapment under an impacting drop , MARIE-JEAN THORAVAL, SIGURDUR T. THORODDSEN, King Abdullah University of Science and Technology (KAUST), Thuwal 23955-6900, Saudi Arabia, KOHSEI TAKEHARA, TAKEHARU GOJI ETOH, Kinki University, Osaka, Japan — We use ultra-high-speed imaging and numerical simulations (GERRIS, http://gfs.sf.net) to observe and analyze the formation of up to 14 air tori when a water drop impacts on a thin liquid film of water or other miscible liquids. They form during the early contact between the drop and the pool by the vertical oscillations of the ejecta sheet. They then break in micro-bubble rings by the Rayleigh instability. Their formation is associated with the shedding of an axisymmetric vortex street into the liquid from the free surface. These vorticity structures and their dynamics are made apparent by the dynamics of the micro-bubbles, added seed particles and the difference of refractive index for different liquids in the drop and the pool. More robust entrapments are observed for a thin film of ethanol or methanol. We show that while the non-spherical drop shape is not responsible for the toroidal bubble entrapments, the number of rings is increasing for more oblate drops. Individual bubble entrapments are also observed from azimuthal destabilizations of the neck between the drop and the pool. [M.-J. Thoraval, K. Takehara, T. G. Etoh, S. Popinet, P. Ray, C. Josserand, S. Zaleski and S. T. Thoroddsen (2012). von Kármán Vortex Street within an Impacting Drop. Phys. Rev. Lett., 108, 264506.]

8:26AM G4.00003 Bouncing and walking droplets , JAN MOLACEK, JOHN BUSH, Massachusetts Institute of Technology — Motivated by the hydrodynamic quantum analogue system of Yves Couder, we examine the dynamics of silicone oil drops bouncing on a vertically vibrating liquid bath. We report regime diagrams indicating the dependence of the vertical drop motion on the system parameters. A logarithmic spring model for the interface is developed, and provides new rationale for the regime diagrams. We further examine the spatio-temporal evolution of the standing waves created on the bath surface by repeated drop impacts. Measurement of the tangential coefficient of restitution of drops bouncing on a quiescent bath enables us to accurately determine all the major forces acting on the drop during flight and impact. By combining the horizontal and vertical dynamics, we thus develop a model for the walking drops that enables us to rationalize both the extent of the walking regime and the walking speeds. The model predictions compare favorably with experimental data in the parameter range explored.

8:39AM G4.00004 A trajectory equation for walking droplets , ANAND OZA, RODOLFO ROSALES, JOHN BUSH, Massachusetts Institute of Technology — Yves Couder and coworkers have demonstrated phenomena reminiscent of quantum mechanics in a macroscopic hydrodynamic system. Specifically, they have discovered that millimetric droplets walking on a vibrating fluid bath exhibit wave-particle phenomena previously thought to be peculiar to the microscopic quantum realm, including single-particle diffraction and tunneling. Orbital quantization may be observed by placing a walking drop on a rotating fluid bath, which suggests a correspondence between the drop's quantized orbits and the Landau levels of an electron in a uniform magnetic field. We here develop an integro-differential trajectory equation for these walking droplets with a view to gaining insight into their subtle dynamics. We present an exact formula for the walking speed and compare it to experimental data. We also analyze the stability of the walking solution to infinitesimal perturbations. The trajectory equation is used to model the walking drop in a rotating fluid bath, which allows us to rationalize the observed orbital quantization. We predict the existence of self-orbiting or "spin" states and a mechanism reminiscent of the Zeeman effect in quantum mechanics.

8:52AM G4.00005 Bouncing droplets in quantized orbits , MATTHIEU LABOUSSE, Langevin Institute, ESPCI ParisTech, STÉPHANE PERRARD, Laboratoire Matière et Systèmes Complexes, UMR 7057 of CNRS and Paris Diderot University, MARC MISKIN, James Franck Institute, University of Chicago, EMMANUEL FORT, Langevin Institute, ESPCI ParisTech, YVES COUDER, Laboratoire Matière et Systèmes Complexes, UMR 7057 of CNRS and Paris Diderot University, JOHN BUSH, Mathematics Department, MIT — A drop of silicon oil on a vibrating bath can surprisingly bounce without coalescing. Slightly below the Faraday threshold, each impact creates a slowly decreasing stationary wave. Driven by the whole past field, a small increment of momentum is given to the droplet at each rebound that leads to a walking regime. This macroscopic wave particle system exhibits fascinating quantum-like behaviour that is strongly experimentally supported by diffraction interference, double slits, Landau levels, tunnel effect, Zeeman effect, cavities. A new step has been taken in the understanding of this system by applying an external potential to a double-structured ferrofluidic drop. Depending of the magnitude of its past field, i.e. the memory of the system, a shift from classical to statistically-quantized behaviours arises. First, we will report the experimental observations of this dual system in the light of the previous experiments. Then, a brief overview of a theoretical approach will be presented to simply rationalize these quantized behaviours.

9:05AM G4.00006 Hydrodynamic quantum analogues: droplets walking on the impossible pilot wave[1] , JOHN BUSH, Department of Mathematics, MIT — Yves Couder and coworkers have demonstrated that droplets walking on a vibrating fluid bath exhibit several features previously thought to be peculiar to the microscopic quantum realm. We explore the connection between this hydrodynamic system and the pilot-wave theory of quantum mechanics proposed by de Broglie and extended by workers in the field of stochastic electrodynamics. Critical common features of these ostensibly disparate systems are identified, and quantitative differences noted.

[1]The author thanks the NSF.

9:18AM G4.00007 Splashing of droplets on liquid surfaces , MICHAEL CHEMAMA, Harvard University, SHREYAS MANDRE, Brown University, MICHAEL BRENNER, Harvard University — High-velocity impacts of liquid droplets on a solid surface produce a splash. This splash is usually accepted to be the consequence of the ejection of a thin sheet of fluid near the impact point. Recent works have shed light on the formation of this liquid sheet by studying the interaction between the droplet and the thin layer of air above the solid surface just before the impact, both experimentally and theoretically. Here we apply the theoretical approach previously applied to splashing on a solid surface [Mandre and Brenner; JFM, 690, 148 (2012)] to the study of splashing on a liquid surface, where the compressed air layer can now deform two liquid interfaces. We show how the nonlinearities in Navier-Stokes equations lead to the ejection of a thin liquid sheet and how this relates to the experimental observations.

9:31AM G4.00008 Numerical investigation of microbubble formation in liquid-liquid impact events[1] , SEYEDSHAHABADDIN MIRJALILI, ALI MANI, Stanford University, Department of Mechanical Engineering — A numerical study of the problem of a droplet impacting another layer of the same liquid is performed with the primary motivation of understanding the steps that lead to the formation of multiple micro-bubbles in the Mesler entrainment mechanism. Simulations start before impact, where a thin gas layer is present and are continued to stages after impact, taking care of topological changes, and finally depicting the formation of the chandelier-like pattern of small bubbles observed in Mesler entrainment. A two dimensional boundary element approach similar to the work of M. Mani, Mandre and Brenner (JFM, vol. 647, p. 163, 2010) is undertaken with the appropriate assumption of inviscid, incompressible potential flow in the liquid bodies, thin structure lubrication flow in the gas layer, with modifications to allow for large interface deflection and topological changes assuming uniform pressure in the bubbles.

[1]Supported by the Office of Naval Research

9:44AM G4.00009 Droplet impact on a liquid pool: air bubble entrainment[1] , TUAN TRAN, University of Twente, Enschede, The Netherlands, HELENE DE MALEPRADE, Ecole Polytechnique, Paris, France, CHAO SUN, DETLEF LOHSE, University of Twente, Enschede, The Netherlands — A range of spectacular phenomena result from impacts of droplets on the surface of a liquid pool, such as splashing, bubble entrapment, or droplet bouncing off from the surface. Here we provide experimental results on the dynamics that precede these striking events, in particular on the entrapment process of an air bubble between the impacting droplet and the pool's surface. We focus on the impact dynamics at early time with an emphasis on the air layer from its formation to its rupture. We identify the dependence of the rupture position on the liquid viscosity and the impact velocity. We show that the volume of the entrapped air under impacting droplets can be related to both that of impacting droplets on solid surfaces and that of impacting rigid spheres on liquid surfaces.

[1]FOM & ERC

9:57AM G4.00010 Drop impact on a non-miscible liquid , HENRI LHUISSIER, CHAO SUN, ANDREA PROSPERETTI, DETLEF LOHSE, University of Twente, POF — The impact of a drop on a deep liquid bath is well-known to transiently open a crater in the bath and possibly eject a liquid sheet and a jet. For non-miscible drop and bath liquids at impact the drop can fragment and disperse into a collection of non-coalescing daughter drops. At impact the drop flattens, spreads at the crater surface and reaches a maximal deformation, which gets larger with increasing impact velocity, before surface tension drives its recession. This recession can promote the fragmentation by two different mechanisms: At moderate impact velocity, the drop recession converges to the axis of symmetry to form a jet which then fragments by a Plateau-Rayleigh mechanism. At higher velocity the edge of the receding drop destabilizes and shapes into ligaments which subsequently fragment. For this later mechanism, the dependence of the critical velocity for fragmentation on the bath viscosity and the number and size distribution of the daughter drops as a function of impact velocity will be discussed.

Monday, November 19, 2012 8:00AM - 10:10AM –

Session G5 Computational Fluid Dynamics IV 24A - Chair: Andrew Cook, Lawrence Livermore National Laboratory

8:00AM G5.00001 Large-eddy simulations of coherent vortices embedded in an impinging jet , WEN WU[1], UGO PIOMELLI[2], Department of Mechanical and Materials Engineering, Queen's University, Kingston (Ontario), Canada — Large-eddy simulations of impinging jets with embedded ring vortices are carried out to study the interaction between the axisymmetric wall jet generated away from the impingement region and the vortex rings. After the primary vortex interacts with the wall, a secondary vortex (with opposite-sign vorticity) is formed from the lifted wall vorticity, which is stretched and wrapped around the primary one. The instability of the secondary vortex dramatically increases the three-dimensionality of the vortex pair. Turbulence statistics reveal a very strong interaction between the two, and intense turbulence generation. The liftup of wall turbulence also results in the generation of rib-like vortices that roll around the primary vortex ring. A three-dimensional, sinusoidal instability of the main vortex is also observed, as well as the generation of near-wall streamwise vortices, which may be related to the striations observed when vortex rings impinge on a sand bed. The stretching and dissipation eventually cause the destruction of the coherent structures of the vortices.

[1]Ph.D candidate, Queen's University.
[2]Professor, Canada Research Chair in Computational Turbulence, HPCVL-Sun Microsystems Chair in Computational Science and Engineering, AIAA Associate Fellow.

8:13AM G5.00002 Numerical simulation of supersonic water vapor jet impinging on a flat plate , KAZUTO KUZUU, Japan Aerospace Exploration Agency (JAXA), JUNYA AONO, Research Center of Computational Mechanics, Inc., EIJI SHIMA, Japan Aerospace Exploration Agency (JAXA) — We investigated supersonic water vapor jet impinging on a flat plate through numerical simulation. This simulation is for estimating heating effect of a reusable sounding rocket during vertical landing. The jet from the rocket bottom is supersonic, M=2 to 3, high temperature, T=2000K, and over-expanded. Atmospheric condition is a stationary standard air. The simulation is base on the full Navier-Stokes equations, and the flow is numerically solved by an unstructured compressible flow solver, in-house code LS-FLOW-RG. In this solver, the transport properties of muti-species gas and mass conservation equations of those species are considered. We employed DDES method as a turbulence model. For verification and validation, we also carried out a simulation under the condition of air, and compared with the experimental data. Agreement between our results and the experimental data are satisfactory. Through this simulation, we calculated the flow under some exit pressure conditions, and discuss the effects of pressure ratio on flow structures, heat transfer and so on. Furthermore, we also investigated diffusion effects of water vapor, and we confirmed that these phenomena are generated by the interaction of atmospheric air and affects the heat transfer to the surrounding environment.

8:26AM G5.00003 A New Formulation for Volume-of-Fluid Simulations of Drops on Solid Surfaces: Inclusion of Adhesion Force , C. CHANG, A. CRISCIONE, S. JAKIRLIC, C. TROPEA, Technical Univ. of Darmstadt, ALIDAD AMIRFAZLI, University of Alberta — The capillary forces acting on a sessile drop placed on a solid surface has two basic components: (1) the Laplace pressure (LP) due to the curvature of the liquid-gas interface, and (2) the Surface Tension Force (STF) as a concentrated force acting at the three-phase contact line. STF can be thought of adhesion force for a drop placed on a solid surface. To date, Volume-of-Fluid (VoF) simulations of drops on solid surfaces have only considered LP, and ignored the STF. Ignoring the STF can lead to incorrect description of the physics for systems involving sessile drops (e.g. shedding of a drop from a surface) especially when capillary and external (e.g. inertial) forces are of the same order of magnitude. Continuum Surface Force (CSF) method is widely used in VoF to model the LP. By modifying the CSF implementation at the contact line, we have added the STF to the VoF formulation. Two case studies, i.e. water drops on an inclined surface and a sessile drop exposed to a shearing airflow are considered. When the STF was ignored, a drop placed on an inclined surface moved at an unrealistically low inclination (e.g. 1 degree for a system with considerable contact angle hysteresis of 10-30 deg.). Same unrealistic motion for the drop was observed when exposed to very low air velocities. Inclusion of the STF corrected both of these unphysical outcomes. A discussion of various systems with different wettabilities (adhesion force values) for each of the two case studies will be provided and comparisons with experiments will be presented.

8:39AM G5.00004 ABSTRACT WITHDRAWN –

8:52AM G5.00005 Turbulent Flows Over Three-Dimensional Shark Skin[1] , AARON BOOMSMA, University of Minnesota, LI WEN, GEORGE LAUDER, Harvard University, FOTIS SOTIROPOULOS, University of Minnesota — Shark skin is covered with thousands of small tooth-like structures called denticles. It has long been hypothesized that denticles act as riblets do in a turbulent boundary layer and help reduce friction drag and enhance shark swimming efficiency. We employ the Curvilinear Immersed Boundary (CURVIB) method (Ge and Sotiropoulos, J. Comp. Physics, 2008) to carry out high-resolution large eddy simulations of turbulent flow past a series of anatomically realistic shark denticles mounted on a flat plate. The denticle shapes used in our simulations were obtained by scanning Mako Short Fin shark skin with micro-CT. The computed results are analyzed to elucidate the three-dimensional structure of the flow past the denticles and identify possible drag reduction mechanics. Drag measurements obtained in a laboratory flume for various denticle spacings and arrangements are also reported and analyzed in tandem with the LES results to explore similarities between shark skin and engineered riblets.

[1]Computational Resources were provided by the University of Minnesota Supercomputing Institute.

9:05AM G5.00006 Three-dimensional simulation of jellyfish by the penalty immersed boundary method , SUNG GOON PARK, CHENG BONG CHANG, HYUNG JIN SUNG, Korea Advanced Institute of Science and Technology, FLOW CONTROL LABORATORY TEAM — The interaction between the motion of a three-dimensional jellyfish and the surrounding fluid was numerically simulated by the penalty immersed boundary method (pIBM). The effects of the vortex formation and the elastic properties on the kinematics of swimming jellyfish were examined. In order to simulate the incompressible fluid motion, the fractional step method was adopted on the Eulerian domain, while the subdivision finite element method was used to describe the solid motion on the Lagrangian domain. Coupling of the fluid motion and the jellyfish motion was realized in the framework of the pIBM. Our results suggest that the starting and stopping vortices, which are respectively induced from a power stroke and a recovery stroke, were formed in the wake of the swimming jellyfish. These two types of vortex interacted with each other, which made the size of vortex larger and caused the augmentation of thrust. Swimming performance of the jellyfish also depended on the elastic properties such as the tension and bending rigidity. It was found that the center velocity of the jellyfish increases with increasing the tension rigidity.

9:18AM G5.00007 Evaluation of Wind Turbine Wake Interaction Models in a RANS Framework[1] , JORDAN WILSON[2], KARAN VENAYAGAMOORTHY[3], Colorado State University — Wind energy produced from horizontal axis wind turbines (HAWTs) remains the most cost effective source of renewable energy production. Computational fluid dynamics (CFD) model studies are widely used as an *a priori* means to study wind farm environments for adequacy of wind resources and optimal configurations. This body of research explores the velocity deficit effect and flow fluctuations created by turbine wakes in a RANS framework for National Renewable Energy Laboratory (NREL) 5MW reference turbines. Various turbine models are explored to determine the most computationally efficient model that accurately captures the physics of interest. While only neutral ABL conditions are simulated in this study, consideration is also given to future work looking at the stable ABL and a full diurnal cycle when selecting a closure model. The objective of this current research is to further understand the development and resolution of turbine wakes for power optimization in neutral ABL conditions with a mind toward fatigue load minimization.

[1]Funded by the Clean Energy Supercluster, CSU.
[2]Graduate Student
[3]Assistant Professor

9:31AM G5.00008 CFD simulation of Urban Environment to study building energy and Urban Heat Island (UHI) implications , NEGIN NAZARIAN, JAN KLEISSL, Mechanical and Aerospace Engineering, University of California San Diego — Numerical simulations are used to study the street-scale urban environment investigating air flow and heat transfer that affect Urban Heat Island formation and urban energy use. Simulations are performed based on Reynolds-averaged Navier-Stokes equations and Large Eddy Simulations using ANSYS/FLUENT. Comprehensive simulations of the daytime urban environment are presented accounting for various contributing factors such as building aspect ratio, stability, and radiative properties of surfaces. Buoyancy and co-occurrence of forced and mixed convective flow regimes are accounted for and the local Richardson number inside the canyon and near building surfaces are examined. A three-dimensional regular building array is used for air flow simulation and thermal analysis. Periodic boundary conditions are used in both stream/span-wise directions representing fully-developed flow and wind profile above the canyon and vortex formation inside the street canyon are studied. The simulations are performed on a clear day in southern California and corresponding daytime solar load is applied for heat transfer purposes. Considering the coupled behavior of thermal effects and flow in the urban environment, we examine surface and canopy air temperature versus building energy use.

9:44AM G5.00009 Large Eddy Simulation of Dilute Sediment Suspension in an Open Channel Flow , GETNET AGEGNEHU, HEATHER D. SMITH, Department of Civil and Environmental Engineering, Louisiana State University — Flow and suspended sediment transport in fully developed turbulent open channel flow has been investigated using Large Eddy Simulation. We used a three-dimensional, non-hydrostatic model, OpenFOAM for this study. Pre-evaluation of three existing turbulence closure schemes is performed by comparing the mean flow and turbulent quantities with the direct numerical simulation results of Moser et al. (1999). It is found that the Dynamic Mixed Smagorinsky model underestimates the wall shear stress compared to the Dynamic Smagorinsky and one equation Eddy Viscosity schemes. Moreover, the Dynamic Smagorinsky scheme gives relatively better results in both the mean and turbulent quantities. The advection-diffusion equation is solved for suspended sediment transport and the effect of sediment roughness is included in the momentum equation based on the rough wall formulation proposed by Cebeci and Bradshaw (1977). A pick up function based on van Rijn (1984) is used to determine the sediment erosion. The settling process is taken into account with a settling velocity appearing in the concentration equation. Sediment and flow quantities are validated by comparing with the experimental data of Lyn (1988). The coupled hydrodynamics results are in good agreement with the experimental data.

9:57AM G5.00010 A three dimensional numerical simulation of current induced sediment processes - A comparison to experimental work , MARKUS BURKOW, Institute for Numerical Simulation, University of Bonn — Current driven sediment transport causes the evolution of bedforms like dunes, ripples or scour marks. These bedforms are formed by the interaction of entrainment and deposition of sediment particles. In this study we use a numerical simulation of the three dimensional fluid flow and the simultaneous transport processes to reproduce these processes. To solve the incompressible two-phase Navier-Stokes equations we use NaSt3D as fluid solver for incompressible flow problems in three dimensions. High order schemes are applied for spatial as well as for temporal discretization. Bed load transport as the main agent responsible for building up bed forms is modelled by the Exner equation. The rearrangement of sediment leads to a new sediment surface height which results in new bedforms. To test our model we compare our results to flume experiments concerning scour erosion at obstacles carried out at Department of Geography, University of Bonn. And as a two-phase example experimental results concerning splash-erosion are compared to our simulations. In both examples simulation and experiment are in good agreement.

Monday, November 19, 2012 8:00AM - 10:10AM –

Session G6 Microfluidics: General I 24B - Chair: Francois Gallaire, EPFL

8:00AM G6.00001 Effect of Secondary Flows on convection-dominated dispersion , ALESSANDRA ADROVER, La Sapienza Università di Roma, ELISABETTA VECA, Centro Ricerca ENEA Casaccia — We investigate the effects of secondary (transverse) flows on "convection-dominated dispersion" of pressure driven, open column laminar flow in a conduit with rectangular cross-section. In the convection-controlled dispersion regime (i.e. laminar dispersion in finite-length channel with axial flow at high Peclet numbers) the properties of the dispersion boundary layer and the values of the scaling exponents controlling the dependence of the moment hierarchy on the Peclet number are determined by the local near-wall behavior of the axial velocity. The presence of transverse flows strongly modify the localization properties of the dispersion boundary layer and consequently the moment scaling exponents. Different secondary flows, electrokinetically induced and independent of the primary axial flow, are considered. A complete scaling theory is presented for the dependence of the n-th order moment of the outlet chromatogram as a function of the axial Peclet number, the secondary flow's pattern and intensity.

8:13AM G6.00002 Geometry-Influenced Slippage on a Bubble Mattress in Microfluidics , ELIF KARATAY, SANDER HAASE, University of Twente, Soft Matter, Fluidics and Interfaces Group, CLAAS WILLEM VISSER, CHAO SUN, DETLEF LOHSE, University of Twente, Physics of Fluids Group, PEICHUN AMY TSAI, ROB LAMMERTINK, University of Twente, Soft Matter, Fluidics and Interfaces Group, SOFT MATTER, FLUIDICS AND INTERFACES GROUP COLLABORATION, PHYSICS OF FLUIDS GROUP COLLABORATION — Hydrodynamic slippage is advantageous for drag reduction and it has been achieved with hydrophobic microstructures. Such substrates can provide soft gas/liquid interfaces with shear-free boundary condition, thereby slippage. The establishment of stable soft-interfaces is crucial for the slippage; however, it has been a challenge. In this study, we design and fabricate hydrophobic microfluidic devices, allowing stable two-phase flow with controllable micro-bubbles at the boundary of the micro-channels. We experimentally and numerically exam the geometric effect of the micro-bubbles on the slippage. The effective slip length is measured for a wide range of protrusion angles, θ, using micro-particle image velocimetry. Our measurements reveal a maximum effective slip length approximately at $\theta =$ 10 degrees. In addition, the experimental results show a decrease in slip length with increasing protrusion angles when $\theta > 10^{\circ}$. The transverse laminar flow over micro-bubbles has also been numerically studied with finite element methods. The experimental results show a good agreement with the numerical results quantitatively.

8:26AM G6.00003 Mass transfer through laminar boundary layer in 2-d microchannels with nonuniform cross section: the effect of wall curvature , AUGUSTA PEDACCHIA, ALESSANDRA ADROVER, La Sapienza Università di Roma — We provide an analytical solution for the combined diffusive and convective 2-d mass transport from a surface film (of arbitrary shape at a given uniform concentration) to a pure solvent flowing in creeping flow conditions into a microchannel, delimited by a flat no-slip surface and by the releasing film itself. Such a problem arises in the study of swelling and dissolution of polimeric thin films under the action of a solvent tangential flow simulating the oral thin film dissolution for drug relase towards the buccal mucosa or oral cavity. We present a similarity solution for laminar forced convection mass (or heat) transfer that generalizes the classical boundary layer solution of the Graetz-Nusselt problem (valid for straight channels or pipes) to a solvent flowing in creeping flow conditions into a 2-d channel with cross-section continuously varying along the axial coordinate x. Close to the releasing boundary, parametrized by a curvilinear abscissa s, both tangential and normal velocity components play a role and their scaling behavior, as a function of wall distance r, should be taken into account in order to have an accurate description of the concentration profile in the boundary layer and of the dependence of the Sherwood number on the curvilinear abscissa s.

8:39AM G6.00004 The Effective Slip Length of a Flow of a Fluid in Cassie State along a Structured Surface , CLARISSA SCHÖNECKER, TOBIAS BAIER, STEFFEN HARDT, Institute for Nano- and Microfluidics, Center of Smart Interfaces, TU Darmstadt, Germany — Microstructured surfaces, like for example superhydrophobic surfaces, can possess a significant apparent slip. This is usually due to a fluid entrapped in the roughness features of the surface. When a second, immiscible fluid flows over such a surface, the presence of the entrapped fluid may lead to a remarkable reduction of drag. So far, the effective slip length of such a flow was only known for a completely dissipation-free fluid being enclosed in the roughness features or for an entrapped fluid which presents a constant local slip length to the outer flow. While the first case completely neglects the viscosity of the enclosed fluid and the geometry of the roughness, the second case lacks the knowledge of the size of the local slip length, besides it being non-constant along a finite interface. We present an analytical expression for the flow field over a surface patterned with rectangular grooves, taking into account dissipation as well as the dimensions of the grooves. This leads to an expression for the effective slip length, which incorporates not only the influence of the viscosity but also provides a direct link between the geometry of the surface structure and the slip length. The results may be of great help for understanding and designing microstructured surfaces.

8:52AM G6.00005 Studying Droplet dynamics by depth-averaged simulation[1] , MATHIAS NAGEL, FRANÇOIS GALLAIRE, Laboratory of Fluid Mechanics and Instabilities - EPFL — Droplets in flat micro channels are deformable, avoid obstacles, interact with one another and undergo separation and coealescence. For the simulation of these micro fluidic two-phase flows we propose a depth-averaged model derived from the Stokes equation, the so called Brinkman equation. This equation is solved by the Boundary Element Method, which leads to a meshless numerical algorithm. A flow solver based on the depth-averaged model computes the dynamic evolution of the droplet in the 2D flow plane and retains the dominant effects in the thin/depth-averaged direction. In addition we developed a set of modified boundary conditions that account for 3D effects like film formation, droplet break-up or capillary action on cavities. We present results for droplet breakup in flow focussing devices and interaction between two droplets. In both cases the numerical results show a good agreement with experiments. The reduction from 3D to 2D by depth averaging and 2D to 1D by transformation of the equations to boundary integrals leads to a significant simplification. Yet the model reproduces essentially the physics to describe these confined two-phase flows.

[1]Project funded by the European Research Council.

9:05AM G6.00006 Creeping three-dimensional flow around a immobile penny-shaped cylindrical droplet , FRANCOIS GALLAIRE, LFMI, EPFL, Lausanne, Switzerland — The flow in a shallow microchannel around a stationary flattened cylindrical viscous droplet at low Reynolds number is considered, using matched asymptotic expansions, with the aspect ratio as small parameter. At leading order, the flow is at rest in the center region of the droplet and it is governed by the two-dimensional Hele-Shaw potential flow equations in the exterior. However, close to the interface, a boundary layer has to be introduced in each fluid, in order to fulfill the kinematic and dynamic boundary conditions. As anticipated from simple scaling arguments, the boundary layer thicknesses scale with the channel height. The next order in the boundary-layer expansion shows the appearance of both radial and cross-plane velocity components. The results are compared to numerical solutions of the full 3-D Stokes equations. Marangoni driving along the interface can be included and yields surprising 3-D recirculation patterns.

9:18AM G6.00007 Taylor-Aris dispersion of droplets (point concentrations) , SØREN VEDEL, EMIL HOVAD, HENRIK BRUUS, Technical University of Denmark — The effective axial diffusion of solute concentrations advected in channel flows known as Taylor-Aris dispersion is caused by the transverse fluid velocity variations present in any channel flow [1,2]. Using our previously developed general theory [3], we study the dispersion of droplets (point concentrations) in steady and unsteady flows. Since the droplet will eventually fill the entire channel, only the transient phase leading up to complete filling requires investigation. Irrespectable of whether the flow is time-dependent or steady, the transient dispersion exhibits a strong dependence on the initial release position, "anomalous" temporal scaling, and surprisingly also shortly exceeds the Taylor-Aris limit. We will show that all these effects, which are unlike the dispersion for transverse uniform initial distributions, are easily understood as being results of variations in the velocity gradient about the release position. This emphasizes that the transient dispersion is caused by the advective stretching of the solute powered by the lateral diffusion, and provides new insight to the underlying mechanisms of Taylor-Aris dispersion for any initial distribution.

[1] Taylor, *Proc. Roy. Soc. Lond. A* **219**, 186 (1953)
[2] Aris, *Proc. Roy. Soc. Lond. A* **235**, 67 (1956)
[3] Vedel and Bruus, *J. Fluid Mech.* **691**, 95 (2012)

9:31AM G6.00008 Does slippage within a superhydrophobic channel always reduce drag? , ANNA LEE, Seoul National University, MYOUNG-WOON MOON, Korea Institute of Science and Technology, HO-YOUNG KIM, Seoul National University — It is commonly perceived that super-hydrophobizing channel walls can reduce drag on liquid flowing inside the channel. Here we point out that rough, hydrophobic channels which induce slippage of liquid flows can exert greater drag than hydrophilic channels exhibiting no slippage of liquid. While air pockets formed between liquid and hydrophobic solid structures allow the liquid to slip over themselves, they also reduce the cross-section of the liquid stream. We theoretically set up a criterion for surface structure that determines whether hydrophobizing or hydrophilicizing textured channel walls would be advantageous in reducing drag on liquid flows. On the basis of our theory, we evaluate the efficiency of previously reported superhydrophobic channels in reducing drag as compared to hydrophilic channels. We also experimentally corroborate our theory by measuring the flow rate of water within microchannels of different surface topography and wettability.

9:44AM G6.00009 Criterion of wetting failure in a Couette flow[1] , PENG GAO, Department of Modern Mechanics, University of Science and Technology of China — Wetting failure occurs when the speed of the moving substrate exceeds a threshold, characterized by a critical Capillary number, above which the stationary contact line cannot be sustained. In most experimental and theoretical studies, it is found that the onset of wetting failure is accompanied by a geometry constraint that the apparent contact angle vanishes. In a Couette device, however, it is reported that wetting failure occurs at nonzero apparent contact angles. Using a lubrication theory, we investigate the contact dynamics in a Couette flow. The critical Capillary number is predicted. It is suggested that the criterion of vanishing apparent contact angle still holds if it is well defined.

[1]This work was supported by the National Natural Science Foundation of China (No. 11102203) and FANEDD (No. 201136).

9:57AM G6.00010 Probing slip boundaries by bubble fingering , HSIEN-HUNG WEI, Department of Chemical Engineering, National Cheng Kung University, YING-CHIH LIAO, Department of Chemical Engineering, National Taiwan University — Motivated by experimental efforts on determining slip length, we propose the Bretherton-type bubble fingering as an alternative strategy for probing slip effects. We find that at sufficiently high bubble speeds (but still in the small capillary number regime), the film thickness obeys classical Bretherton's 2/3 law. However, when the bubble speed is below some critical value where the film thickness is comparable to the slip length, a new quadratic law will emerge to govern the behavior of the film thickness below the slip length. The critical bubble speed is also found to be proportional to the 3/2 power of the slip length. In the analogous thermocapillary problem, a bubble in a slip channel can travel much faster than in a no-slip channel, at the speed proportional to the 5/3 power of the slip length. Effects of disjoining pressure and surfactant are also discussed and the results also show strong dependence on the slip length.

Monday, November 19, 2012 8:00AM - 10:10AM –
Session G7 Suspensions II 24C - Chair: Salima Rafai, Grenoble University

8:00AM G7.00001 Cahn-Hilliard modeling of particles suspended in two-phase flows , PATRICK ANDERSON, YOUNG JOON CHOI, Eindhoven University of Technology — We study the dynamics of particles suspended in two-phase flows by coupling the Cahn-Hilliard theory with the extended finite element method (XFEM). In the Cahn-Hilliard model the interface is considered to have a small but finite thickness, which circumvents explicit tracking of the interface. For the direct numerical simulation of particle-suspended flows, we incorporate an XFEM, in which the particle domain is decoupled from the fluid domain. To cope with the movement of the particles, a temporary ALE scheme is used for the mapping of field variables at the previous time levels onto the computational mesh at the current time level. The model is general, but to demonstrate and validate the technique, here the dynamics of a single particle at a fluid-fluid interface is studied. First, we apply a small disturbance on a particle resting at an interface between two fluids, and investigate the particle movement towards its equilibrium position. In particular, we are interested in the effect of interfacial thickness, surface tension, particle size and viscosity ratio of two fluids on the particle movement towards its equilibrium position. Finally, we show the movement of a particle passing through multiple layers of fluids.

8:13AM G7.00002 Effect of the History Force on Particle Trajectories within an Oscillatory Rotating Flow , SHUJING XU, ALI NADIM, Claremont Graduate University — At a previous APS-DFD meeting it was reported, based on theoretical considerations, that particles denser than their suspending fluid can be made to migrate *toward* the rotation axis if the container undergoes oscillatory rigid-body rotation in an appropriate range of frequencies [Nadim et al., *Bull. Am. Phys. Soc.*, **53**, 191 (2008)]. This is contrary to ordinary centrifugation. However, the effect of the Basset history force was not accounted for in that analysis. It is shown here that while the history force significantly affects the dynamics of the particles, the oscillatory "counter-centrifugation" effect that was previously discovered continues to persist even when the history force is included in the analysis. Interestingly, inclusion of the history force can extend the parameter regime for which oscillatory counter-centrifugation might be observed.

8:26AM G7.00003 Taylor-Aris dispersion in the presence of shear-enhanced diffusion and variable mean flow , GREGORY RUBINSTEIN, IVAN CHRISTOV, HOWARD STONE, Princeton University — Controlling the dispersion of colloidal suspensions is important in applications ranging from drug delivery to water purification. Previously, Griffiths and Stone [EPL (2012) 97, 58005] considered the influence of shear-induced diffusion on the Taylor-Aris dispersion of a colloidal suspension flowing in a cylindrical pipe. In this work, we extend their analysis to a radial outflow geometry, which features velocity variations along the flow direction. We found that the shear-induced diffusion due to the hydrodynamic interactions between the colloidal particles tends to decrease dispersion in the flow direction, as does the decrease in the velocity as the fluid flows radially outward. Using the method of multiple time scales, we derived an averaged dispersion equation that demonstrates the impact of these two effects. We also extended our methodology to coupled dispersion problems, in which the suspended particulate phase releases heat into the ambient fluid or the colloidal particles dissolve into the solvent medium.

8:39AM G7.00004 Dispersion of suspension plugs in microchannels , MARTIN MAXEY, GLAREH AZADI, ANUBHAV TRIPATHI, Brown University, KYONGMIN YEO, Lawrence Berkeley National Laboratory — Plugs of beads can be used as a model for understanding the behavior of suspensions in microfluidic devices. Despite a large volume of literature in the field of microfluidic suspension flows; the fundamental understanding of the dispersion of hard and soft particle suspension plugs of finite length has not be studied in detail. Here we focus on the dispersion characteristics of non-Brownian, low Reynolds number bead suspensions in microfluidic channels. The effect of initial plug length, bead size and dilution on the early stages of dispersion of these beads in a pressure driven flow will be presented. Numerical simulations of suspension shear flow for test configurations relevant to plug dispersion have been performed in order to present a new level of continuum models for particle stresses and particle fluxes in confined shear flows. These results can contribute to the design of high throughput microfluidic systems for cell screening and bio-separation.

8:52AM G7.00005 Hydrodynamics of confined suspensions , NICOLAS DESREUMAUX, ESPCI, University Pierre et Marie Curie, RAPHAEL JEANNERET, ESPCI, JEAN-BAPTISTE CAUSSIN, ENS Lyon, ERIC LAUGA, UCSD, DENIS BARTOLO, ESPCI — We investigate experimentally and theoretically the dynamics of suspensions flowing in quasi-bidimensional channels. Specifically, we focus on the role of the hydrodynamic interactions on the large-scale behavior of the suspensions. Our experiments demonstrate that density fluctuations propagate on large distances with a non-linear dispersion relation. We then model our system as a collection of interacting potential dipolar singularities. By focusing on the large-scale dynamics, we show that density waves do propagate and that the dispersion relation depends on the local curvature of the density field. This model yields good agreement with our experiments.

9:05AM G7.00006 Hydrodynamic particle interactions in sheared microflows , ALVARO MARIN, MASSIMILIANO ROSSI, Institute of Fluid Mechanics and Aerodynamics, Bundeswehr University Munich, MAURICIO ZURITA-GOTOR, Abengoa Research, Sevilla, Spain, CHRISTIAN J. KÄHLER, Institute of Fluid Mechanics and Aerodynamics, Bundeswehr University Munich — Multiphase flows in micro-confined geometries are non-trivial problems: drops and particles introduce a high degree of complexity into the otherwise linear Stokes flows. Very recently, new mechanisms of instability have been identified in simulations in shear-flows of non-Brownian particle solutions (Zurita-Gotor et al., J. Fluid Mech. 592, 2007, and Phys. Rev. Lett. 108, 2012), which might be the cause for anomalous self-diffusion measured experimentally by Zarraga and Leighton (Phys. Fluids 14, 2002). Using a 3D particle tracking technique (Astigmatism-PTV), we perform experiments in a microconfined cone-plate couette flow with a dilute suspension of non-brownian particles. The A-PTV technique permits us to track individual particles trajectories revealing particle-particle hydrodynamic interactions. Our experiments show an abnormal dispersion in the velocity field and non-homogeneous particle distribution which can be related with the swapping mechanism (JFM 592, 2007;PRL 108, 2012).

9:18AM G7.00007 The Effect of Particle Deformation on the Collective Dynamics of Confined Rigid Spheres and Deformable Drops[1] , J. BLAWZDZIEWICZ, Texas Tech University, M. LOEWENBERG, Yale University, P.J.A. JANSSEN, SABIC Innovative Plastics, M.D. BARON, Princeton University, P.D. ANDERSON, Eindhoven University of Technology, E. WAJNRYB, IPPT, Warsaw, Poland — The evolution of linear arrays of rigid spheres and deformable drops in a Poiseuille flow between parallel walls is investigated to determine the effect of particle deformation on the collective dynamics in confined particulate flows. We find that linear arrays of rigid spheres aligned in the flow direction undergo a particle-pairing instability and are unstable to lateral perturbations. Linear arrays of deformable drops, in addition to the pairing instability, exhibit other dynamical features, including formation of transient triplets, cascades of pair-switching events, and formation of pairs with equal interparticle spacing. Particle deformation also stabilizes drop arrays to lateral perturbations. These pairing and alignment phenomena are qualitatively explained in terms of hydrodynamic far-field dipole interactions (insensitive to particle deformation) and quadrupole interactions (deformation induced). We suggest that quadrupole interactions underlie the spontaneous formation of droplet strings in confined emulsions under shear [*Phys. Rev. Lett.*, 2001, **86**, 1023].

[1]This work was supported by NSF grant No. CBET 1059745.

9:31AM G7.00008 Shape asymmetric particles self-assemble under flow in quasi-two-dimensional microchannels , WILLIAM USPAL, Department of Physics, Massachusetts Institute of Technology, H. BURAK ERAL, PATRICK DOYLE, Department of Chemical Engineering, Massachusetts Institute of Technology — Emerging applications in microfluidics increasingly require the imposition of spatial and temporal order on flowing suspensions of particles. In on-chip flow cytometry, for instance, cells must be individually distinguishable and addressable as they flow through a scanning region. Via combined theoretical and experimental approaches, we consider how particle shape can be tailored for flow induced assembly in a shallow, "quasi-two-dimensional" microchannel. Our main finding is that when fore-aft symmetry is broken, a single rigid particle will spontaneously align with the external flow field and migrate laterally to the channel centerline. Via a simple theoretical model, quantitatively borne out by experiments, we show how assembly arises from the interplay of lateral confinement by side walls and a particle's hydrodynamic self-interaction. This mechanism is unique to the quasi-two-dimensional channel geometry: strong confinement in one spatial direction and weak confinement in another. Moreover, assembly does not require time reversal symmetry breaking, as is commonly supposed. Building on this understanding of a single particle, we show that clusters of multiple asymmetric particles likewise assemble into spatially ordered, "crystalline" states.

9:44AM G7.00009 Rheological measurements in liquid-solid mixtures , ESPERANZA LINARES-GUERRERO, MELANY HUNT, Caltech — In a previous investigation by Koos *et al.* (2012), the torque measurements of mixtures of non-cohesive, neutrally buoyant, relatively large particles (order of mm diameter size) in aqueous glycerine at high shear rates were presented. These measurements showed a linear dependence on the range of shear rates tested ($\dot{\gamma} \approx 1 - 110 s^{-1}$) and a nonlinear dependence on the solid fraction. For this range of shear rates, previous studies have shown a transition from a linear to a nonlinear dependence on $\dot{\gamma}$. However, most of these studies considered smaller particle size (the order of μm) and therefore small Stokes number, or the measurement devices were dominated by secondary flows as shown by Hunt *et al.* (2002). To analyze further this rheology, experiments for suspensions with particle densities higher than the liquid density were performed, which increased the Stokes number. To avoid slip at the walls, the rheometer walls were roughened. At higher Stokes number the particle collisions become important and change the rheology of the suspension. Preliminary results show an increased dependence on Stokes number compared with the earlier study.

9:57AM G7.00010 Particle Migration and Interaction in Confined Flows , KAITLYN TULEY, SUNGYON LEE, MARCUS ROPER, UCLA — Inertial microfluidic principles are widely applied in flow cytometry, microfluidic chips, and cell filtration. The hydrodynamic nonlinearity caused by inertia regulates the location of and separation between particles in these devices. Yet, there is no theory to explain these phenomena. We describe asymptotic and numerical models for the fundamental fluid mechanics of particle migration and interaction with applications to: (i) inertial focusing and (ii) the dynamic self-assembly of particles into uniformly spaced flowing lattices.

Monday, November 19, 2012 8:00AM - 9:44AM –

Session G8 Drops V 25A - Chair: Paul Steen, Cornell University

8:00AM G8.00001 Surface Patterns of Parametrically-Excited Sessile Drops , CHUN-TI CHANG, JOSHUA BOSTWICK, SUSAN DANIEL, PAUL STEEN, Cornell University — A mechanically-excited sessile water drop exhibits surface patterns that vary with driving amplitude and frequency. At small amplitudes, drops exhibit axisymmetric wave patterns. At sufficiently large amplitudes and at particular frequencies, symmetry breaks and there appear subharmonically-resonating sectoral and tesseral mode shapes. We report observations from experiment and compare to results from modeling the parametric excitation of the drops. The modeling uses the spectrum and eigenmodes from a linear stability analysis of the inviscid sessile Rayleigh drop with moving/pinned contact-line. Manipulating drop motion can be important to a variety of applications.

8:13AM G8.00002 Sessile Rayleigh drop instability , PAUL STEEN, Cornell University, JOSH BOSTWICK, NC State University — Rayleigh (1879) determined the mode shapes and frequencies of the inviscid motion of a free drop held by surface tension. We study the inviscid motions of a sessile Rayleigh drop – a drop which rests on a planar solid and whose contact-line is free to move. Linear stability analysis gives the modes and frequencies of the droplet motions. In this talk, we focus on the "walking instability," an unstable mode wherein the drop moves across a planar substrate in an inviscid rocking-like motion. The mode shape is non-axisymmetric. Although the experimental literature has hinted at such a mode, this is the first prediction from linear stability analysis, as far as we are aware. The "walking instability" of the drop converts energy stored in the liquid shape into the energy of liquid motion – which represents a heretofore unknown pathway of energy conversion of potentially wide significance for a broad range of applications.

8:26AM G8.00003 Motion and coalescence of sessile drops driven by substrate wetting gradient and external flow , MAJID AHMADLOUYDARAB, JAMES J. FENG, Chemical & Biological Engineering Department - UBC — We report finite-element simulations of the motion and coalescence of sessile drops driven by solid substrate wetting gradient and external flow. When the external flow and wetting gradient favor motion in opposite directions, their competition determines the behavior of a single sessile drop. If two drops are placed on a solid substrate with wetting gradient, the trailing drop may catch up with the leading one and coalesce with it. This is owing to the greater viscous friction on the leading drop, which is more spread out on a more hydrophobic area of the substrate. An external flow alone can induce coalescence, thanks to differential drags on the two drops. When both wetting gradient and flow effects coexist and compete, more complex scenarios arise, with either coalescence or separation depending on the strength of each.

8:39AM G8.00004 Measuring the force of drag on air sheared sessile drops , ANDREW J.B. MILNE, BRIAN FLECK, ALIDAD AMIRFAZLI, University of Alberta — To blow a drop along or off of a surface (i.e. to shed the drop), the drag force on the drop (based on flow conditions, drop shape, and fluid properties) must overcome the adhesion force between the drop and the surface (based on surface tension, drop shape, and contact angle). While the shedding of sessile drops by shear flow has been studied [Milne, A. J. B. & Amirfazli, A. *Langmuir* 25, 14155 (2009).], no independent measurements of the drag or adhesion forces have been made. Likewise, analytic predictions are limited to hemispherical drops and low air velocities. We present, therefore, measurements of the drag force on sessile drops at air velocities up to the point of incipient motion. Measurements were made using a modified floating element shear sensor in a laminar low speed wind tunnel to record drag force over the surface with the drop absent, and over the combined system of the surface and drop partially immersed in the boundary layer. Surfaces of different wettabilities were used to study the effects of drop shape and contact angles, with drop volume ranged between approximately 10 and 100 microlitres. The drag force for incipient motion (which by definition equals the maximum of the adhesion force) is compared to simplified models for drop adhesion such as that of Furmidge [Furmidge, C. G. L., *J. Colloid Sci.* 17, 309 (1962).]

8:52AM G8.00005 Drop Stability on Leaves[1] , WILL BLACKMORE, YONGKANG CHEN, CHRIS HINOJOSA, RYAN JENSON, DUC NGUYEN, ANDREW WOLLMAN, MARK WEISLOGEL, Portland State University — The stability of drops on surfaces has implications to many natural and industrial processes. Critical configurational drop stability is defined herein as any change in a control parameter (i.e. drop volume) that leads to movement to a new location on the surface and/or detachment from the surface. The configurational stability of 'wall-bound drops" is enhanced by contact line pinning at sharp edges. In this work an extensive array of computations are performed for "wall-edge-vertex bound drops" (a.k.a. drops on blade tips or drops on leaf tips which they resemble). The numerical approach applies the Surface Evolver algorithm through implementation of a new file layer and a multi-parameter sweep function. As a consequence, thousands of critical drop configurations are efficiently computed as functions of contact angle, blade edge vertex half-angle, and g-orientation. Simple experiments are performed to benchmark the computations which are then correlated for ease of application. It is shown that sessile, pendent, and wall-edge bound drops are only limiting cases of the more generalized blade-bound drops, and that the ubiquitous "dry leaf tip" is observed for a range of the critical geometric and wetting parameters.

[1]NASA NNX09AP66A: GRC, NASA NNX10AK68H: Oregon Space Grant Consortium

9:05AM G8.00006 ABSTRACT WITHDRAWN –

9:18AM G8.00007 The shape of a drop between two rigid fibers , SUZIE PROTIERE, CNRS/Institut Jean le Rond d'Alembert, CAMILLE DUPRAT, HOWARD A. STONE, MAE-Princeton University — Wetting of fibrous media is observed in many engineered systems, e.g. filters, textiles, paper etc. and may also be found in Nature (e.g. hair or feathers). To understand the basic response of such material when interacting with a liquid we study the model system of a finite volume of liquid on two parallel rigid fibers. A liquid wetting the fibers can adopt two distinct equilibrium shapes: a compact hemispherical drop shape or a long liquid column of constant cross-section. These two morphologies depend on the inter-fiber distance, the liquid volume, the fiber radius and the liquid-fiber contact angle. We study the transitions between a drop shape and a column by incrementally varying the inter-fiber distance and find that the transition depends on the global geometry of the system as well as on the volume of liquid. More surprisingly we find that these two morphological states may coexist for certain parameter values. These switches in morphologies may be used to manipulate or transport liquid at a small scale.

9:31AM G8.00008 A closed-form analytical solution for both hanging and sitting droplets[1] , JUAN MANUEL GOMBA, Insitute of Physics Arroyo Seco, UNCPBA - Argentina, CARLOS ALBERTO PERAZZO, Dept. Phys. and Chem, University Favaloro - Argentina — We present a new analytical solution for the shape of both hanging and sitting droplets under the effects of gravity, surface tension and molecular forces arising between the liquid and the substrate. These molecular London/van der Waals and electrostatics forces, which are described by means of potential functions, are responsible for the existence of a nanometric precursor film that surrounds the droplet. The analytical solution describes pancake-type profiles and also droplets elongated in the vertical direction. We find novel expressions that relate microscopic and nanoscopic aspects, such as the strength of the molecular forces and the thickness of the nanometric precursor film, with macroscopic quantities, e. g., the cross sectional area, the height and the width of the droplet. Note: No pancakes will be served.

[1]CONICET / ANPCyT

Monday, November 19, 2012 8:00AM - 10:10AM –

Session G9 Interfacial/Thin Film Instability III 25B - Chair: Devin Conroy, Imperial College London

8:00AM G9.00001 Electrosprays: cone-jet breakup in the presence of DC electric fields[1] , OMAR MATAR, DEVIN CONROY, RICHARD CRASTER, DEMETRIOS PAPAGEORGIOU, Imperial College London, HSUEH-CHIA CHANG, Notre Dame University — The breakup of an electrified jet in a gas with an axially applied electric field is investigated theoretically. The jet fluid is taken to be a symmetric electrolyte and modelling of the cationic and anionic species is carried out by considering the Nernst-Planck equations in order to find the volume charge density that influences the electric field in the jet. At high flow rates, the governing equations are investigated asymptotically in the long-wave limit and the one-dimensional model is solved numerically for a wide range of hydrodynamic, electrostatic, and electrokinetic parameters. For low flow rates, a boundary integral method is used to solve for the electrostatic potential surrounding the interface, which assumes a cone-like shape from which a thin jet emanates. In both cases, the electric field causes the jet to stretch and thin to a point where ion repulsion forces the jet to undergo Rayleigh fission. We measure the axial distance at which this point occurs by comparing the jet radius to the distance at which ion repulsion is important.

[1]Royal Academy of Engineering Distinguished Visitor Fellowship for Prof. Chang, and EPSRC

8:13AM G9.00002 On the minimum size of drops composing the type of monodisperse microemulsions obtained via tip streaming , JOSE MANUEL GORDILLO, Universidad de Sevilla, ALEJANDRO SEVILLA, Universidad Carlos III, ELENA CASTRO-HERNANDEZ, Universidad de Sevilla — On a recent paper, Castro et al (JFM (2012), 698, 423-445, JFM12) reported the generation of concentrated monodisperse emulsions composed of drops with sizes even below 1 μm. Drops are generated from the capillary breakup of a long and thin ligament which strongly stretches downstream from the exit of an injector of radius R_i. The ligament is formed when a flow rate Q_i of a fluid with a viscosity μ_i discharges into an immiscible liquid of viscosity μ_o flowing in parallel with the axis of the injector at a velocity qU_0, with $q = Q_i/(\pi R_i^2 U_o)$. It was theoretically found that the diameter of the drops obtained is $D_i \propto R_i q^{1/2}$. However, experiments showed that the predicted drop size is only found when the highly stretched ligament is formed. But this occurs for values of $\lambda = \mu_i/\mu_o$ and the capillary number $Ca_o = \mu_o U_o/\sigma$, with σ the interfacial tension coefficient, above a certain threshold which depends on the flow rate ratio q. In this presentation we theoretically show that the boundaries in the (Ca_o,λ,q) space in which highly stretched long ligaments are formed, corresponds to the conditions under which the jet, calculated using the slender-body description of JFM12, is *globally stable*.

8:26AM G9.00003 Simulating the capillary breakup of a liquid torus , HADI MEHRABIAN, JAMES J. FENG[1], Department of Chemical and Biological Engineering, University of British Columbia — Capillary instability of a Newtonian liquid torus suspended in a bath of Newtonian liquid is computed using Cahn-Hilliard diffuse interface method. The main difference between the torus and a straight thread is the presence of curvature and flow filed caused by the shrinkage of the torus. We show that the capillary wave initially grows linearly as on a straight thread. The azimuthal curvature decreases the growth rate while the flow field enhances the growth rate of the capillary waves. The initially dominant mode does not necessarily persist into nonlinear growth and eventual breakup. Breakup depends on the competition of two timescales: torus retraction and neck pinchoff. We demonstrate that it is determined by the initial amplitude, the aspect ratio and the viscosity ratio of the torus viscosity ratio. The numerical results are in agreement with experimental results.

[1]Department of Mathematics, University of British Columbia

8:39AM G9.00004 Marangoni convection in a thin film: Formation of a fractal hierarchy of droplets , ARTHUR STRAUBE, Department of Physics, Humboldt University of Berlin, Germany, ANTON ASHMANOV, Perm State University, Russia, SERGEY SHKLYAEV, Institute of Continuous Media Mechanics, Perm, Russia, ARKADY PIKOVSKY, Department of Physics and Astronomy, University of Potsdam, Germany — A thin liquid film heated from below is known to demonstrate rupture due to the longwave Marangoni convection [J. Fluid Mech. 345, 45 (1997)]. Despite a number of numerical and theoretical studies nothing has been known about the final state that sets in as a result of the instability. In our recent one-dimensional analysis [Phys. Rev. E 82, 020601 (2010)] we have shown that the terminal state is a fractal built of a hierarchy of droplets, each of which can be represented as a dissipative compacton. The latter can be thought of as the stationary droplet with the zero contact angle and whose profile is determined by the interaction of the surface tension and thermocapillary flow. We show that the dimension of a set of the compactons is equal to unity or, equivalently, the dimension of gaps between the droplets is zero. For a generalized axisymmetric problem, two types of compactons are found: hat-like and ring-like ones, with the maximum either at the symmetry axis or at a finite radius, respectively. Again, an initially flat film becomes unstable, forming a hat-like compacton and a fractal sequence of ring-like compactons, whose stability is analyzed with respect to the perturbations breaking the axial symmetry.

8:52AM G9.00005 The effect of charge regulation on the stability of electrolyte films , CHRISTIAAN KETELAAR, VLADIMIR AJAEV, Southern Methodist University — The stability of a thin liquid film of an electrolyte on a solid substrate is investigated. We show that the commonly used approximation of constant charge densities at the solid-liquid and solid-gas interfaces does not lead to predictions of film rupture. To reconcile the model with experimental observations, the effects of charge regulation are incorporated into the model using a linear relationship between charge density and potential. Linear stability criteria are formulated in terms of charge regulation parameters and electrolyte properties. A nonlinear evolution equation for film thickness is derived and solved numerically over a range of parameters to determine the conditions of film rupture.

9:05AM G9.00006 Self-similar rupture of thin heated viscous sheets , M. BOWEN, Waseda University, B.S. TILLEY, Worcester Polytechnic Institute — We consider the evolution and rupture dynamics of a thin viscous planar sheet subject to a symmetric initial disturbances in the thermal and velocity fields. We consider the long-wave limit where deviations from the mean sheet velocity are small, but thermocapillary stresses, fluid inertia, van der Waals effects, capillarity, and heat transfer to the environment can be significant. The result is a coupled system of three equations that describe the sheet thickness, the sheet velocity, and the sheet temperature. When van der Waals effects are dominant, the sheet ruptures due to disjoining pressures for sufficiently long-wave disturbances on a faster time-scale than convection or conduction. However, in cases when disjoining pressures are small, we find a self-similar rupture process where inertia, viscous stresses, thermocapillarity, convection and conduction all balance. We quantify how solutions transition from this similarity solution to the van-der-Waals-driven self-similar solution when the thickness of the sheet becomes sufficiently thin.

9:18AM G9.00007 Capillary structures formed with viscous threads in microchannels[1] , SAMIRA DARVISHI, THOMAS CUBAUD, Stony Brook University — We investigate two-fluid flows with highly viscous threads formed within a sheath of a less viscous liquid. An experimental study is conducted in long plane microfluidic chambers to examine the formation of periodic capillary structures that result from the viscous folding instability. For the case of a non-wetting thread, a phase diagram of flow patterns is presented including thread breakup, partially coalesced folds, and fully coalesced folds regimes. When the thread is partially wetting, a range of forced wetting and thin film instabilities is observed during the thread lubrication transition in the cell. Novel combinations of viscous and capillary instabilities are shown to offer innovative mechanisms for manipulating the lubrication properties of high-viscosity fluids in microsystems.

[1]This work is supported by NSF (CBET-0932925)

9:31AM G9.00008 Application of Stress Jumps in Free Surface Films for Noise-Free and Controlled Growth of 3D Microarrays , NAN LIU, SANDRA TROIAN, California Institute of Technology, MC 128-95, Pasadena, CA 91125 — Linear stability analysis of a nanoscale, free surface viscous film exposed to a large and uniform transverse thermal gradient has revealed how a flat interface deforms spontaneously into a large area, periodic array consisting of elongated protrusions. These formations, however, are extremely sensitive to noise and exhibit substantial variation in position, shape and growth rate, unacceptable for film patterning strategies based on fluid instability. In this talk, we demonstrate how patterned thermal fields can be used to induce large amplitude stress jumps which help better localize periodic formations and trigger more rapid growth. In particular, we present results of finite element simulations in which waveform trains develop in response to thermocapillary stress jumps induced by an adjacent cooled protrusion with sharp sidewalls. We compare the shapes and growth rates of two different regions of the film, one grown by linear instability and the other by a large amplitude stress jump. In general, stress jumps lead to more rapid and regular formation of pillar arrays insensitive to noise. Despite the large amplitude perturbation, the wavelength remains close to the value predicted by linear stability, a significant advantage for technological applications.

9:44AM G9.00009 Dynamics of a moving liquid sheet in the presence of acoustics[1] , MAHESH TIRUMKUDULU, MANJULA PARAMATI, Department of Chemical Engineering, Indian Institute of Technology Bombay, Mumbai, India, PETER SCHMID, Laboratorie d'Hydrodynamique, Ecole Polytechnique, Paris, France — A moving liquid sheet produced by impinging two collinear jets of water was recently shown to be unstable to a select set of acoustic frequencies (Mulmule et al, Vol 22, 022101, Phys. Fluids, (2010)). In order to better understand the phenomenon, we have developed a non-contact technique based on laser induced fluorescence to measure both the displacement and the thickness variation of the liquid sheet. As expected, the liquid sheet thickness varies inversely with radial distance from the point of impingement in the absence of acoustics. In the presence of acoustic forcing (50-150 Hz), experiments reveal that while the sinuous deformation mode is present at all frequencies, the sheet break at the select set of frequencies occurs due to significantly high growth rates of the varicose mode. We present the variation of the measured wave speeds and growth rates as a function of the forcing frequency. These results agree qualitatively with the predictions of the aforementioned study which shows that the varicose and sinuous modes are coupled at the lowest order when the sheet is subjected to acoustics.

[1]Acknowledment: Indo-French Centre for the Promotion of Advance Research, and IIT Bombay

9:57AM G9.00010 Thin Film Behavior Under External Vibrations , MICHAEL BESTEHORN, QI HAN, Dep. Theoretical Physics, BTU Cottbus, Germany — We study the dynamics of a thin liquid film on a horizontal or weakly inclined substrate. The film is parametrically excited by mechanical vertical and horizontal oscillations. Inertia effects are taken into account and the standard thin film formulation is extended by a second equation for the mean flow rate

$$q(x,t) = \int_0^{h(x,t)} u(x,z,t)\, dz \ ,$$

where h is the film thickness and u the horizontal velocity. The set of two coupled PDEs for h and q allows for resonances and instabilities of the flat film due to external vibrations. Linear results based on a damped Mathieu equation as well as fully nonlinear results in the frame of longwave approximation found numerically will be presented. For certain regions in the amplitude-frequency plane as well as for certain forms of the excitations we obtain standard Faraday patterns like oscillating squares, but also hexagons and much more involved spatial and temporal pattern formation are observed.

Monday, November 19, 2012 8:00AM - 10:10AM –

Session G10 Instability: Jets, Wakes and Shear Layers IV 25C - Chair: Matthew Juniper, University of Cambridge

8:00AM G10.00001 Resonant Oscillations of Shallow Flow Past a Cavity: Exchange Coefficients and Depthwise Variations , BURAK AHMET TUNA, EGEMEN TINAR, DONALD ROCKWELL, Lehigh University — Fully turbulent shallow flow past a cavity can give rise to highly coherent oscillations, which arise from coupling between: the instability of the separated layer along the cavity; and a standing gravity wave mode within the cavity. These oscillations yield large increases in the time-averaged entrainment and mass exchange coefficients between the cavity and the main flow. Such increases are due to substantial enhancement of turbulent stresses in the separated shear layer during the coupled oscillation, relative to the stresses associated with no coupling. Patterns of the flow structure have been characterized as a function of elevation above the bed (bottom surface) of the shallow flow. At elevations close to the bed, the time-averaged streamlines are deflected inwards towards their center of curvature. This streamline deflection is due to radial migration of flow along the bed, which arises from a radial pressure gradient. In addition, patterns of normal and shear Reynolds stresses are substantially altered as the bed is approached. These changes of stresses with depth are, in turn, associated with degradation of coherent, phase-averaged patterns of vortex formation in the separated shear layer. All of the foregoing aspects of the depthwise variations of the flow patterns are related to corresponding changes of the entrainment and mass exchange coefficients.

8:13AM G10.00002 Shallow Flow Past a Cavity: Self-Excited Oscillations due to Resonant Coupling , MAXWELL WOLFINGER, DONALD ROCKWELL, CEM OZEN, Lehigh University — A fully turbulent shallow flow past a cavity can give rise to highly coherent oscillations. Coupling between the instability of the separated shear layer along the cavity and a gravity standing wave mode within the cavity results in: sharp spectral peaks of fluctuating pressure along the cavity wall; and substantial modification of the flow patterns along and within the cavity. Onset of the fully coupled, highly coherent oscillation of the shear layer-cavity system occurs as follows. As the inflow velocity along the cavity increases, the instability frequency of the separated shear layer approaches the frequency of the gravity standing wave mode. When these frequencies are coincident, the instability frequency locks-on to (remains the same as) the standing wave frequency, and highly ordered, time-dependent deflections of the free-surface occur. The peak amplitude of the unsteady pressure fluctuation occurs during this locked-on state. Moreover, quantitative imaging in the form of particle image velocimetry reveals large-scale vortex formation in the separated shear layer, which is associated with substantial changes of time- and phase-averaged flow patterns within the cavity. In turn, these features of the flow are associated with large increases of Reynolds stresses in the separated layer along the cavity.

8:26AM G10.00003 LES Investigation of instabilities in cavity flow with a top boundary , AARTHI SEKARAN, GERALD MORRISON, Texas A&M University — The effect of the influence of a top boundary on cavity instabilities is studied using Large Eddy Simulations (LES). The motivation for the geometry was the flow over the cavities of a hole-pattern seal, where sudden changes in instability modes could lead to large variations in the rotordynamic stability of the system. A single, two-dimensional cavity is modeled at different conditions to study the occurrence of phenomenon such as the shear layer instability and the wake mode instability. The simulations are successfully able to capture both modes and this is verified via a spectral analysis of the data. The study also details the occurrence and development of each instability mode and discusses its physical effect on the overall flow behavior in the system. Qualitative comparisons are then made with cavities without top boundaries in order to determine particular differences due to the presence of the same.

8:39AM G10.00004 ABSTRACT WITHDRAWN –

8:52AM G10.00005 Kelvin-Helmholtz instability in a confined geometry , PAUL BONIFACE, MSC Laboratory - University Paris 7 Diderot and Saint-Gobain Recherche, LUC LEBON, MATHIEU RECEVEUR, MSC Laboratory - University Paris 7 Diderot, FABIEN BOUILLET, Saint-Gobain Recherche, LAURENT LIMAT, MSC Laboratory - University Paris 7 Diderot — Growth of kelvin-Helmholtz instabilities received many attention in the case of wakes or shear layers with thickness increasing in the downstream direction. In contrast with these "unstationary" situations, few works investigate stationary situations in which a Kelvin Helmholtz raw grows and saturates inside an imposed geometry. One of the sole exception is an experiment developed by M. Rabaud & Y. Couder where vortices develop between concentric rotating disks. We developed an experiment in a different geometry: a recirculating belt is running at the free surface of a long rectangular tank of larger width. In some conditions a regular rectilinear raw of vortices develop between the flow dragged by the belt and the flow recirculating beside the belt. When the belt is on the central axis of the tank, with free surface on each side, recirculation flows on each side. In this case two regular raws can develop. Vortices of these two raws can interact and a 180° phase shift between them is observed. Properties of these vortices have been investigated and we tried to relate it to available models. Parallels with jet instability can be made. We also have explored the occurrence condition of the recirculation: at low reynolds number it can occur via the bottom of the tank.

9:05AM G10.00006 Evolution of inviscid Kelvin-Helmholtz instability from a piecewise linear shear layer , ANIRBAN GUHA, MONA RAHMANI, GREGORY LAWRENCE, University of British Columbia — Here we study the evolution of 2D, inviscid Kelvin-Helmholtz instability (KH) ensuing from a piecewise linear shear layer. Although KH pertaining to smooth shear layers (eg. Hyperbolic tangent profile) has been thorough investigated in the past, very little is known about KH resulting from sharp shear layers. Pozrikidis and Higdon (1985) have shown that piecewise shear layer evolves into elliptical vortex patches. This non-linear state is dramatically different from the well known spiral-billow structure of KH. In fact, there is a little acknowledgement that elliptical vortex patches can represent non-linear KH. In this work, we show how such patches evolve through the interaction of vorticity waves. Our work is based on two types of computational methods (i) Contour Dynamics: a boundary-element method which tracks the evolution of the contour of a vortex patch using Lagrangian marker points, and (ii) Direct Numerical Simulation (DNS): an Eulerian pseudo-spectral method heavily used in studying hydrodynamic instability and turbulence.

9:18AM G10.00007 3D optimal perturbations developing in homogeneous mixing layers in presence of subharmonic vortex-pairing , ADRIANA LOPEZ-ZAZUETA, LAURENT JOLY, JEROME FONTANE, Universite de Toulouse, ISAE, DAEP — Many experimental and numerical studies have found that the pairing of primary Kelvin-Helmholtz (KH) vortices in mixing layers generally inhibits the growth of infinitesimal three-dimensional disturbances, delaying the transition to turbulence. In this work, we investigate the existence of 3D perturbations that grow fast enough to survive the subharmonic merging instability. For this purpose, we perform a numerical study of the transient linear evolution of 3D perturbations emerging in a homogeneous time-evolving mixing layer which undergoes pairing. We look for the optimal perturbation that yields to the largest gain of energy at a specific time horizon, by the use of an optimization method which solves iteratively the linearized direct and adjoint Navier-Stokes equations. In particular, we consider the influence of the time horizon relative to the saturation times of both the primary KH and the subharmonic pairing instabilities.

9:31AM G10.00008 Methodologies for solving Vortex Wave Interaction problems to obtain edge states , ANDREA ISONI, Imperial College London, HUGH BLACKBURN, Monash University, PHILIP HALL, SPENCER SHERWIN, Imperial College London — The interaction of waves with streamwise vortices (Vortex Wave Interaction Theory or VWI Theory) is relevant in motivating the "self-sustained processes" and in delineating perturbations to shear flows which may become either laminar or turbulent. It has been recognised that a streamwise vortex velocity field (U,V,W) can be decomposed into an O(1) axial U-component, "streak" field, and a O($R^{-1}*$) roll field (V,W). As identified by Hall and Smith 1991, an equilibrium solution can be produced by an interaction of the non-linear wave terms with the roll field within the critical layer. VWI theory has been investigated numerically on a Couette flow using three different approaches, which we refer to as asymptotic, regularised and hybrid methods. In the first approach, the roll equations are subjected to jump conditions along the critical layer as proposed by Hall and Smith 1991. In the second approach, a body forcing, which regularises the jump conditions, is added on the roll equations as discussed in Hall and Sherwin 2010. In the third approach, a forcing term proportional to the divergence of the Wave Reynolds Stresses is imposed on the roll equations. In this presentation we will discuss the merits of each of these approaches and the connection with the lower branch solution.

9:44AM G10.00009 Identifying instability mechanisms in swirling shear flows by using all components of the structural sensitivity[1] , MATTHEW JUNIPER, UBAID QADRI, University of Cambridge — Four different physical mechanisms can cause or support instability in swirling shear flows (Gallaire and Chomaz 2003, PoF 15(9) 2622-2639). These are: axial shear, inertial waves, centrifugal instabilities, and azimuthal shear. In relatively simple flows, such as a Rankine vortex with plug axial flow, analytical methods can identify the physical mechanisms active in each region of the flow. In more complex flows, such as a vortex breakdown bubble, analytical methods cannot be applied and, in any case, regions of the flow are not easily delineated. When considering the stability of perturbations on top of a base flow, the structural sensitivity quantifies the effect of altering the feedback between the perturbation velocity vector and the perturbation momentum equation. We examine the nine components of this structural sensitivity, firstly for simple flows such as solid body rotation, secondly for complex swirling flows. The first analysis identifies the signature of each physical mechanism, such as the Kelvin-Helmholtz instability and the Coriolis mechanism. The second analysis compares these signatures with those found in different regions of the complex swirling flows. In this way, we identify the physical mechanisms that are active in each region of the more complex flow.

[1]Supported by the European Research Council and by Trinity College Cambridge

9:57AM G10.00010 Experimental Study of LES Models in Turbulent Stratified Flows , DUO XU, JUN CHEN, School of Mechanical Engineering, Purdue University — Stratification caused by density difference leads to significant changes of flow structure. To achieve accurate and realistic results of stratified turbulent flows in large-eddy simulation (LES), the the behavior of subgrid-scale (SGS) models is crucial. In this study, two-dimensional high resolution velocity and scalar (density) dataset from a turbulent stratified jet, obtained through applying a combined Particle Image Velocimetry (PIV) and Planar Laser Induced Fluorescence (PLIF) technique, is used to study the behavior of SGS models. A strong correlation between SGS stress and scalar flux is observed. In particular, variations of turbulent Prandtl number in stable stratification and unstable stratification regions are explored from the experimental data.

Monday, November 19, 2012 8:00AM - 9:57AM –

Session G11 Bubbles II 26A - Chair: Sadegh Dabiri, University of Notre Dame

8:00AM G11.00001 Motion of a bubble ring in a viscous fluid , JING LOU, MING CHENG, Institute of high Performance Computing, Singapore, T.T. LIM, National University of Singapore, INSTITUTE OF HIGH PERFORMANCE COMPUTING, SINGAPORE TEAM, NATIONAL UNIVERSITY OF SINGAPORE TEAM — A Lattice Boltzmann Method and limited experiments were undertaken to study the dynamic of a vortex ring bubble (or bubble ring) in a viscous incompressible fluid. The study is motivated partly by our desire to assess whether a bubble ring keeps increasing its radius and decreasing its propagation velocity as it rises through fluid was predicted by Turner (1957) and Pedley (1968) or does the ring eventually reach a steady state where its radius and velocity remain constant as was predicted by Joseph et al (2008). The parameters investigated included ring circulation, Reynolds number, density ratio and Bond number. Our experimental and numerical results show that a rising bubble ring increases its radius and decreases its velocity, but the process is interrupted by ring instability that eventually causes it to break up into smaller bubbles. For the range of flow conditions investigated, there is no evidence that a bubble ring has attained a constant speed and constant radius before breaking up. Furthermore, it is found that increasing initial circulation has a stabilizing effect on a bubble ring while increasing Reynolds number or Bond number hastens ring instability, resulting in an earlier break up into smaller bubbles.

8:13AM G11.00002 Rising of Taylor Bubble through a Liquid-Liquid Interface , TEE TAI LIM, RAJEEV JAIMAN, HUANG SHA, National University of Singapore — A gas bubble moving through liquids in a round tube can exhibit various behaviors of both theoretical and practical interest. When the tube diameter is not too large, the large bubbles (i.e., Taylor bubbles) are smooth and glossy, with a bullet-shaped nose, and they rise at a constant velocity along the axis of the tube. The present study aims at investigating the rising of a Taylor bubble through a liquid-liquid interface both experimentally by using high-speed video camera and numerically through the volume-of-fluid (VOF) approach based on the adaptive-refinement. When a bubble risers in a liquid and eventually through a liquid-liquid interface, the interfacial force is transferred to the buoyancy and viscous forces. The layer of the heavy surrounding liquid between the bubble and the interface (termed as thin-film) prevents the bubble from reaching the light phase instantaneously. After some time, significant viscous forces, the pressure gradient within the thin-film acts to drain and eventually the film becomes exceedingly small; and bubble achieves a higher terminal rise velocity and the bubble is elongated. The preliminary results obtained by the VOF approach are in qualitative agreement with the experiment.

8:26AM G11.00003 Modeling the drainage of viscous bubbles , CASEY BARTLETT, MATTHIEU SANTIN, JAMES BIRD, Boston University — The lifespan of viscous thin film bubbles are largely dictated by the drainage dynamics of the film. For large enough bubbles, these dynamics are driven by gravity and regulated by viscosity. Past models have assumed that these forces lead to a drainage velocity that increases monotonically with increasing angle from the center axis. Here we show alternative solutions more consistent with experimental data where drainage velocity is not monotonic. We use a combination of numeric and analytic approaches to determine the evolution of the film drainage and investigate if such evolution can be approximated with a self-similar profile. Finally we compare our model results to recent experimental data.

8:39AM G11.00004 Measuring axisymmetric drainage of large viscous bubbles , JAMES BIRD, CASEY BARTLETT, MATTHIEU SANTIN, Boston University — Large bubbles on the surface of a viscous liquid can be stable for many minutes, even in the absence of surfactants. Over time the thickness of these bubbles evolve as the liquid film drains under the influence of gravity. Past interferometry measurements have shown that the film thickness at the top of a viscous bubble decays exponentially – which is consistent with current theories. However these models rely on drainage assumptions away from the centerline, assumptions that have been yet to be validated experimentally. In this talk we present measurements for both the film thickness and drainage velocity along viscous bubbles. Our results demonstrate that current models dramatically under-predict the film thickness away from the centerline. We demonstrate why the dynamics are more subtle than previously assumed, and we offer a model that is consistent with our measurements.

8:52AM G11.00005 DNS of rising bubbles in a vertical homogeneous shear flow , SADEGH DABIRI, University of Notre Dame — The bubbly flow occurs in many natural and industrial situations such as boilers and bubble column reactors. In many of these flows, bubbles rise inside a shear layer. Interaction between bubbles and the shear creates a lateral lift force on the bubbles and affects their distribution in the domain which in turn will affect the drag force on the flow and the flow rate. Here, the rising motion of buoyant bubbles in a homogeneous shear flow in vertical direction is studied. In order to create a homogeneous shear flow, periodic boundary condition in all three directions is implemented. A finite difference method with front tracking is developed that satisfies the periodic-shear boundary condition. The effect of the deformability of bubbles on the magnitude and direction of the lateral lift force is discussed.

9:05AM G11.00006 Size-differentiated lateral migration of bubbles in Couette flow of two-dimensional foam , HADI MOHAMMADIGOUSHKI, JAMES J. FENG, Department of Chemical and Biological Engineering, University of British Columbia — In this Talk, we report experiments on lateral migration of bubbles in a two-dimensional foam sheared in a narrow-gap Couette device. A larger bubble in an otherwise monodisperse bubble raft migrates toward the center of the gap as long as the bubble size ratio and the shear rate are each above a threshold. The migration speed is roughly two orders of magnitude higher than that of a single bubble, and increases with the shear rate and the size ratio. The bubble also deforms much more than an isolated one at the same shear rate. Modifying the Chan-Leal solution for the migration of a single submerged bubble or drop, we derive a formula that successfully predicts all the migration trajectories recorded in the experiment. The threshold for migration corresponds to the wall repulsion force overcoming the capillary force in the 2D foam. The size-differentiated bubble migration provides an explanation for previously observed size segregation in sheared 3D polydisperse foams.

9:18AM G11.00007 Numerical simulations of bubble dynamics at high Reynolds numbers[1] , SAUL PIEDRA, EDUARDO RAMOS, Center for Energy Research-UNAM, TERMOCIENCIAS TEAM — We present a three-dimensional numerical simulation of air bubbles rising in water. The analysis is based on the solution of the conservation equations combined with a front tracking method to represent an interface between two immiscible fluids. The interfacial forces incorporate the effect of the surface tension and the material properties of the fluids are calculated in the entire integration domain. In order to follow the bubbles along thousands of diameters in its ascending motion, a moving reference frame technique is used. The shape of the bubbles, the pressure and the velocity fields at different flow conditions calculated with our model are in agreement with experimental observations reported in the literature. Also, the qualitative change in the trajectory of the bubbles from rectilinear to zig-zag to helical motion is reproduced by the model. Dominant physical effects in each mode of displacement are described.

[1]S.P. acknowledges support from CONACYT-Mexico through a PhD grant

9:31AM G11.00008 Bubble Transport through Micropillar Arrays[1] , KENNETH LEE, University of California, Berkeley, OMER SAVAS, University of California at Berkeley — In current energy research, artificial photosynthetic devices are being designed to split water and harvest hydrogen gas using energy from the sun. In one such design, hydrogen gas bubbles evolve on the 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. Therefore, an efficient method of collecting the evolved gas bubbles is crucial. Preliminary flow visualization has been conducted of bubbles advecting through dense arrays of pillars. Bubbles moving through square and hexagonal arrays are tracked, and the results are qualitatively described. Initial attempts to correlate bubble motion with relevant lengthscales and forces are also presented. These observations suggest how bubble transport within such pillar arrays can be managed, as well as guide subsequent experiments that investigate bubble evolution and collection.

[1]This material is based upon work performed by the Joint Center for Artificial Photosynthesis, a DOE Energy Innovation Hub, supported through the Office of Science of the U.S. Department of Energy under Award Number DE-SC0004993.

9:44AM G11.00009 Bubbles dancing in a vortex: trapping air at a T-junction , DANIELE VIGOLO, NATHAN TYRELL, Princeton University, STEFAN RADL, Graz University of Technology, HOWARD STONE, Princeton University — We present an unusual phenomenon that occurs to low density material, and in particular air bubbles, entrained in a fluid when flowing through a T-junction. For a range of Reynolds numbers, the flow develops two symmetric vortices. Air bubbles are forced to the center of the vortex due to the centrifugal force and, for Reynolds number, Re, greater than ≈ 220, are then "trapped", i.e. they accumulate inside the vortex. Bubbles eventually oscillate (i.e. "dance") in the vortex when the flow becomes unsteady for $Re > 550$. Experiments were conducted by generating H_2, O_2 or simply air bubbles in the range $Re = 100$ to $\approx 6,000$ in a variety of T-junction devices. We have also observed a size dependence of the trapping phenomenon. In addition, our 3D numerical simulations have revealed a gradient of pressure, similar to vortex breakdown, that drives the flow towards the center of the T-junction creating two recirculating zones, which trap air bubbles. The presence of light material or air trapped in a flow could be relevant to industrial systems and biological flows, such as blood vessels, and may contribute to unexpected complications and/or failures in these systems.

Monday, November 19, 2012 8:00AM - 10:10AM –

Session G12 Vortex IV 26B - Chair: Peter Hamlington, University of Colorado

8:00AM G12.00001 Listening to the horseshoe vortex system: interpretation of turbulent coherent structures by Parameter Mapping Sonification , JUAN-PABLO CACERES, Stanford University, Pontificia Universidad Catolica de Chile, CRISTIAN ESCAURIAZA, Pontificia Universidad Catolica de Chile — The adverse pressure gradient induced by a surface-mounted obstacle in a turbulent boundary layer causes the formation the dynamically rich horseshoe vortex system around the body. Recent studies have identified the complex mechanisms responsible for the dynamics of the vortices and the emergence of bimodal histograms of velocity fluctuations in the junction region. To understand the dynamic relation of the multiple vortices, we convert streamwise velocity time-series at the symmetry plane into sound by Parameter Mapping Sonification, to make emerge aspects of the rich-dynamics of the turbulent coherent structures in the vortex system that may not have been uncovered by traditional methods. Through this development we provide insights on the analysis of turbulent flows dominated by the quasi-periodic interaction of large-scale coherent vortices.

8:13AM G12.00002 A flow visualization and volumetric particle-image velocimetry study on low Reynolds number inclined rectangular jets , YONG CHUAN KHOO, TSI Instruments Singapore Pte Ltd, TZE HOW NEW, Nanyang Technological University of Singapore, WING LAI, TSI Incorporated — Flow visualizations and three-dimensional volumetric particle image velocimetry measurements were performed on Re=2,500 jets produced by aspect-ratio of three rectangular nozzles inclined at 60° along their major and minor-planes. Results show that persistently inclined azimuthal ring vortices are formed, regardless of the exact incline-plane used. When the nozzle is inclined along its major-plane, these ring vortices undergo rapid realignments which reduce their inclination and coherence. In contrast, they retain their inclination for a significantly further downstream distance when the nozzle is inclined along its minor-plane instead. As such, gross large-scale vortex structures and behaviour are relatively similar to those observed for inclined elliptic nozzles previously. In contrast to inclined elliptic nozzles with no corners, small-scale streamwise vortex structures are produced by the rectangular nozzle sharp corners which hasten the onset of large-scale vortical changes for a faster transition to flow incoherence. These observations indicate that at the present working conditions, inclined jet behaviour is more sensitive towards the redistribution of jet circulation caused by the nozzle inclination, rather than variations between different nozzle base geometries.

8:26AM G12.00003 Ellipsoidal vortices beyond the quasi-geostrophic approximation , YUE-KIN TSANG, DAVID DRITSCHEL, JEAN REINARD, University of St Andrews — As a model of geophysical vortices, we study numerically the behavior of an ellipsoidal volume of uniform potential vorticity in a non-hydrostatic, rotating stratified flow. Previous studies on ellipsoidal vortices mainly focus on the quasi-geostrophic regime. Here, we aim to determine the effects of ageostrophy on the stability, evolution and the final fate of the vortices. Cyclonic and anti-cyclonic vortices of various aspect ratios are investigated. Generally, cyclonic vortices with roughly circular cross section tend to be more stable. In the cases when the vortices become unstable, a wide range of nonlinear evolution is observed including tumbling, shape vacillation, splitting and filaments shedding. Diagnostics concerning the vortex rotation rate and the degree of "imbalance," which quantifies the amount of internal gravity waves emission, will be presented.

8:39AM G12.00004 On Optimal Vortex Structures for Palinstrophy Generation , DIEGO AYALA, BARTOSZ PROTAS, McMaster University — We are interested in identifying the vortex structures which lead to the largest growth of palinstrophy $\mathcal{P}$ in flows governed by the 2D Navier–Stokes equation in a periodic domain. This problem is a 2D counterpart of the problem concerning the maximal production of enstrophy $\mathcal{E}$ in 3D Navier–Stokes flows which is inherently related to the question of finite–time singularity formation. We investigate the sharpness of the following analytic estimates

$$\frac{d\mathcal{P}}{dt} \sim \mathcal{P}^2 \quad \text{and} \quad \max_{t>0} \mathcal{P}(t) \sim \mathcal{P}(0)^2 \quad \text{as} \quad \mathcal{P} \to \infty$$

by studying suitable optimization problems. These problems are solved for different values of the palinstrophy using modern methods of PDE–constrained optimization combined with DNS. In regard to the instantaneous problem, we discover two families of maximizing fields characterized by distinct properties. We also present evidence that the maximum growth of $\frac{d\mathcal{P}}{dt}$ with $\mathcal{P}$ is in fact weaker than suggested by the analytic estimate.

8:52AM G12.00005 A highly adaptive three dimensional hybrid vortex method for inviscid flows , DAN LUCAS, University of Bristol, DAVID G. DRITSCHEL, University of St Andrews — Motivated by outstanding problems surrounding vortex stretching (e.g. explosive vorticity growth, equation regularity and associated nonlinear depletion), a new hybrid vortex method for inviscid, three-dimensional, incompressible flows is presented. Special emphasis on spatial adaptivity is given to resolve as broad a range of scales as possible in a completely self-similar fashion. We discretise vorticity in Lagrangian filaments (space curves) and compute velocity on an adapted finite-volume grid. This allows for a two-fold adaptivity strategy. First, although naturally spatially adaptive by definition, the vorticity filaments undergo "renoding"; nodes are redistributed along the filaments to concentrate their density in regions of high curvature. Secondly the Eulerian mesh is adapted to follow high strain by determining resolution by way of local filament dimensions. These features allow vortex stretching and folding to be resolved in a completely automatic and self-similar way, in addition the filaments present themselves as a candidate for novel flow diagnostics. Validation of the method is demonstrated via relatively recently discovered helical vortex equilibria (Lucas & Dritschel 2009).

9:05AM G12.00006 2D FTLE in 3D Flows: The accuracy of using two-dimensional data for Lagrangian analysis in a three-dimensional turbulent channel simulation , MATTHEW ROCKWOOD, MELISSA GREEN, Syracuse University — In experimental, three-dimensional vortex-dominated flows, common particle image velocimetry (PIV) data is often collected in only the plane of interest due to equipment constraints. For flows with significant out of plane velocities or velocity gradients, this can create large discrepancies in Lagrangian analyses that require accurate particle trajectories. A Finite Time Lyapunov Exponent (FTLE) analysis is one such example, and has been shown to be very powerful at examining vortex dynamics and interactions in a variety of aperiodic flows. In this work, FTLE analysis of a turbulent channel simulation was conducted using both full three-dimensional velocity data and modified planar data extracted from the same computational domain. When the out of plane velocity component is neglected the difference in FTLE fields is non-trivial. A quantitative comparison and computation of error is presented for several planes across the width of the channel to determine the efficacy of using 2D analyses on the inherently 3D flows.

9:18AM G12.00007 Numerical simulation of the velocity field and vorticity in a flow occurring in a channel and an open domain with periodic forcing[1] , ERICK LOPEZ-SANCHEZ, GERARDO RUIZ-CHAVARRIA, Facultad de Ciencias Universidad Nacional Autonoma de Mexico — The system under study is a flow with periodic forcing in a channel that flows out toward an open domain. At the channel exit a pair of vortices is formed for each period. Using a pseudo-spectral method based on Chebyshev polynomials, the Navier-Stokes and continuity equations are solved numerically in the stream function-vorticity formulation (2D). The equations are integrated during several periods and for different values of Reynolds and Strouhal numbers. We found that the dipoles persist for more than one period and that their evolution depends strongly on the Strouhal number as predicted in previous works (e. g. Dynamics of Atmospheres and Oceans, 37 (2003) 223-244) but a rich dynamics emerges when interaction among vortices is taken account. For instance coalescence of vortices of the same sign can be observed. Besides this case, other two possibilities can be envisaged, namely, the dipoles travel a long distance if compared with the channel width and escapes or the dipoles fails to form completely and return to the channel when period is so short. Otherwise, numerical simulation shows the appearance of a sinusoidal instability which is a mechanism for the destruction of the vortices. Comparison with experimental and observational data is performed giving a good agreement.

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

9:31AM G12.00008 A Bathtub Vortex under the Influence of a Taylor Column in a Rotating Tank[1] , SHIH-LIN HUANG, YIN-CHUNG CHEN, ZI-YA LI, CHIN-CHOU CHU, CHIEN C. CHANG, National Taiwan University — Numerical simulations and laboratory experiments were conducted to investigate a bathtub vortex under the influence of a Taylor column in a rotating tank. A central drain hole is placed at the bottom of the tank and a top-down cylinder is suspended from the rigid lid. We examine the effects of the Rossby number, *Ro* and the Ekman number, *Ek*. Steady-state solutions are shown to have good agreements with flow visualizations and PTV measurements. It is found that at *Ro*$\sim 10^{-2}$, a bottom Ekman pumping forms a classic one-celled structure for the case of no suspended cylinder h/H=0, while for various $h/H \neq 0$, the strong interaction of the bathtub vortex and Taylor column results in a two-celled structure with an inner Ekman pumping and an outer Taylor column induced upwelling. In $h/H \neq 0$, the Taylor wall separates the vortex into an inner and an outer region, but allows the outer fluid to flow into the inner region through a top and a bottom gap which can be classified into two and three flow paths, respectively. Moreover, the individual flow rate of each path and the weaker influence of the Taylor column at *Ro*$\sim$1 and $\sim 10^2$ are also discussed. Finally, we observe that the vorticity strength of the vortex exhibits the relationship with a dimensionless group $\sqrt{fQ/gH^2(1-h/H)}$.

[1]NSC 98-2221-E-002-094-MY3; NSC 99-2111-M-002-005-MY2

9:44AM G12.00009 Unbounded Immersed Interface solver, also for use in Vortex Particle-Mesh methods , YVES MARICHAL[1], PHILIPPE CHATELAIN, GREGOIRE WINCKELMANS, Universite catholique de Louvain (UCL), Institute of Mechanics, Materials and Civil Engineering (iMMC) — We present a new and efficient algorithm to solve the 2-D Poisson equation in unbounded domain and with complex inner boundaries. It is based on an efficient combination of two components: the Immersed Interface method to enforce the boundary condition on each inner boundary (here using solely 1-D stencil corrections) and the James-Lackner algorithm to compute the outer boundary condition consistent with the unbounded domain solution. The algorithm is here implemented using second order finite differences and is particularized to the computation of potential flow past solid bodies. It is validated, by means of grid convergence studies, on the flow past multiple bodies (some also with circulation). The results confirm the second order accuracy everywhere. The algorithm is self consistent as "all is done on the grid" (thus without using a Vortex Panel boundary element method in addition to the grid). The next aim of this work is then to integrate this algorithm in the Vortex Particle-Mesh (VPM) method for the computation of unsteady viscous flows, with boundary layers, detached shear layers and wakes. Preliminary results of the combined methods will also be presented.

[1]Research Fellow (PhD student) of the F.R.S.-FNRS of Belgium

9:57AM G12.00010 Buoyancy-Induced Columnar Vortices[1] , MARK SIMPSON, ARI GLEZER, Georgia Institute of Technology — The formation of anchored, buoyancy-driven columnar vortices driven by the instability of a thermally stratified air layer, and sustained by entrainment of ground-heated air is investigated in a meter-scale laboratory facility. In hot-climate regions, 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 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 the 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 columnar vortex using stereo-PIV with emphasis on scaling and assessment of the available kinetic energy. It is shown that the strength and scaling 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 the Georgia Tech Research Corporation.

Monday, November 19, 2012 8:00AM - 10:10AM –

Session G13 Geophysical: General IV 27A - Chair: Alberto Scotti, University of North Carolina

8:00AM G13.00001 Erosion sculptures , LEIF RISTROPH, M.N.J. MOORE, STEPHEN CHILDRESS, MICHAEL SHELLEY, JUN ZHANG, New York University, Courant Institute — Erosion by flowing fluids carves the striking landscapes imprinted on the Earth and on the surfaces of our neighboring worlds. In these processes, solid boundaries both influence and are shaped by the surrounding fluid, but the emergence of morphology as a result of this interaction is not well understood. We study the coevolution of shape and flow in the context of clay bodies immersed in fast flowing water. Although commonly viewed as a smoothing process, we discover that erosion sculpts surprisingly sharp points and corners that persist as the body shrinks. These features result from a natural tendency to form surfaces that erode uniformly, and we argue that this principle may also apply to the more complex scenarios that occur in nature.

8:13AM G13.00002 Simulation of Sediment Wave Generation and Maintenance , GARY HOFFMANN, ECKART MEIBURG, MOHAMAD NASR-AZADANI, UC Santa Barbara — Deep-sea sediment waves are a common feature throughout the world, forming under the influence of turbidity currents, thermohaline currents, and/or deformational processes. Past efforts for modeling turbidity-current generated sediment waves have focused on 1D (depth-averaged) Navier-Strokes equations, whereas here we use the 2D equations, as implemented in TURBINS 2D. We employed two experimental setups: 1) Repeated flows in a lock-exchange configuration, in which deposition and erosion are observed to lead to waveforms developing on an initially linear ramp, and 2) single continuous inflow currents that are allowed to flow over a pre-existing sinusoidal geometry. Using these two setups, we examine both the initial generation of sediment waves over the course of many episodic events, as well as maintenance of sediment waves under the influence of a quasi-steady flow. In both setups, we examined a range of flow parameters, such as ramp length, sediment settling velocity, etc.

8:26AM G13.00003 Self-similarity of an eroding body , M. NICK J. MOORE, LEIF RISTROPH, STEPHEN CHILDRESS, MICHAEL SHELLEY, Courant Institute, JUN ZHANG, New York University Department of Physics, Courant Institute, APPLIED MATH LAB TEAM — Motivated by the erosion of natural landforms, we study the interaction between an eroding body and surrounding fluid-flow using experiments and simulations. Both reveal the emergence of a unique shape that forms early and then shrinks in a self-similar fashion. Here, I focus on simulations in which erosion rate is dictated by local shear stress. In this high-Reynolds-number context, we determine shear stress by combining an outer, inviscid flow with a boundary layer flow. We discover that a broad range of initial shapes morph into a terminal shape characterized by nearly uniform shear.

8:39AM G13.00004 Numerical simulation of turbulence and sand-bed morphodynamics in natural waterways under live bed conditions[1] , ALI KHOSRONEJAD, FOTIS SOTIROPOULOS, Saint Anthony Falls Laboratory, University of Minnesota — We develop and validate a 3D numerical model for coupled simulations of turbulence and sand-bed morphodynamics in natural waterways under live bed conditions. We employ the Fluid-Structure Interaction Curvilinear Immersed Boundary (FSI-CURVIB) method of Khosronejad et al. (Adv. in Water Res., 2011). The mobile channel bed is discretized with an unstructured triangular grid and treated as the sharp-interface immersed boundary embedded in a background curvilinear mesh. Transport of bed load and suspended load sediments are combined in the non-equilibrium from of the Exner-Poyla for the bed surface elevation, which evolves due to the spatio-temporally varying bed shear stress and velocity vector induced by the turbulent flow field. Both URANS and LES models are implemented to simulate the effects of turbulence. Simulations are carried out for a wide range of waterways, from small scale streams to large-scale rivers, and the simulated sand-waves are quantitatively compared to available measurements. It is shown that the model can accurately capture sand-wave formation, growth, and migration processes observed in nature. The simulated bed-forms are found to have amplitude and wave length scales ranging from the order of centimeters up to several meters.

[1]This work was supported by NSF Grants EAR-0120914 and EAR-0738726, and National Cooperative Highway Research Program Grant NCHRP-HR 24-33. Computational resources were provided by the University of Minnesota Supercomputing Institute

8:52AM G13.00005 Spectral description of migrating bedforms and sediment transport , MICHELE GUALA, NICHOLAS BADHEARTBULL, ARVIND SINGH, EFI FOUFOULA-GEORGIOU, St Anthony Falls Laboratory, Dep. Civil Engineering, NCED, UMN — The spatio-temporal evolution of migrating bedforms in an experimental straight flume with erodible bottom and fixed side-banks is presented through a joint spectral analysis of surface elevations in the frequency and wave number domains. In this framework any generic bedform can be described as a combination of Fourier modes propagating with well defined, scale-dependent, convection velocities. Quasi-simultaneous measurement of bed elevations z=f(x,t) in time (t) and space (x, along the flow direction) are used to estimate the propagation velocity of each Fourier mode, and to propose a purely statistical, spectral approach to quantify sediment transport rates in the presence of bedforms. A functional relationship between the length scale and time scale of migrating bedforms is developed into a dimensionless expression for scale-dependent convection velocities. The latter is further used to provide an estimate of sediment transport rate in the presence of generic migrating bedforms. Experiments were conducted in the Tilting Bed Flume at SAFL under varying flow and bed material conditions.

9:05AM G13.00006 Entraining gravity currents , CHRIS JOHNSON, ANDREW HOGG, School of Mathematics, University of Bristol — Large-scale gravity currents, such as those formed when industrial effluent is discharged at sea, are greatly affected by the entrainment and mixing of ambient fluid into the current, which both dilutes the flow and causes an effective drag between the current and ambient. We study these currents theoretically by combining a shallow-water model for gravity currents flowing under a deep ambient with an empirical model for entrainment, and seek long-time similarity solutions of this model. We find that the dependence of entrainment on the bulk Richardson number plays a crucial role in the current dynamics, and results in entrainment occurring mainly in a region close to the flow front, reminiscent of the entraining current 'head' observed in natural flows. The long-time solution of an entraining lock-release current is a similarity solution of the second kind, in which the current grows as a power of time that is dependent on the form of the entrainment model, approximately as $t^{0.44}$. The structure of a current driven by a constant buoyancy flux is quite different, with the current length growing as $t^{4/5}$. Scaling arguments suggest that these solutions are reached only at very long times, and so may be attained in large natural flows, but not in small-scale experiments.

9:18AM G13.00007 Numerical simulation of reversing buoyancy gravity currents , SENTHIL RADHAKRISHNAN, UC Santa Barbara, ERIK LENK, Westsaechsische Hochschule Zwickau, MICHAEL BOEKELS, RWTH Aachen, ECKART MEIBURG, UC Santa Barbara — Sediment laden fluid propagates as an underflow when its bulk density is higher than the density of the ambient fluid. If the density of the interstitial fluid in gravity current is smaller than the density of the ambient fluid, the gravity current can become positively buoyant after sufficient particles have settled. The current then lifts off from the bottom surface and travels as a surface gravity current over the heavier ambient fluid. These types of currents, where the buoyancy reverses its direction, have been observed when sediment laden fresh water enters the sea or during volcanic eruption that creates a pyroclastic flow. We use a lock-exchange configuration with mono-disperse and bi-disperse particles to study the lofting characteristics of reversing buoyancy currents. This talk will focus on results obtained from Large-eddy Simulation of high Reynolds number currents. In particular, the deposit profiles show a sharp decay at the lift-off point unlike a ground hugging turbidity current whose deposit profile has a slow monotonic decay from the lock region.

9:31AM G13.00008 Power-law for gravity currents produced from instantaneous sources propagating on inclined boundaries in the deceleration phase[1] , ALBERT DAI, Tamkang University — The power-law for gravity currents on slopes is essentially an equivalent form of the solution of thermal theory, when the gravity current is sufficiently far into the deceleration phase. However, the hypothesis that gravity current is sufficiently far into the deceleration phase is hardly satisfied in experiments. In this paper, we re-derived the power-law, considering the influence of bottom friction, and corrected an error in an early version of power-law given by Maxworthy (2010). When the gravity current is not sufficiently far into the deceleration phase, we showed that the power-law still robustly describes the front location versus time relationship, but the amount of heavy fluid in the head can be easily underestimated. The underestimation of heavy fluid in the head also depends on where the gravity current is in the deceleration phase. Therefore, a correction factor is suggested according to the location of gravity current.

[1]Supported by National Science Council in Taiwan

9:44AM G13.00009 Laminar flow of constant-flux released gravity currents: Friction factor-Reynolds number relationship , FIRAT TESTIK, NAZLI YILMAZ, MIJANUR CHOWDHURY, Clemson University — This study aims to provide a relationship for the friction factor, C_f, in terms of the Reynolds number, *Re*, for two-dimensional constant-flux release gravity currents during viscous-buoyancy propagation phase. Motivation of this study was related to the pipeline disposal of high-concentration dredged fluid-mud. Such disposal operations form non-Newtonian gravity currents that propagate over the coastal seafloor. Our theoretical and experimental analysis resulted in C_f–*Re* relationships for both Newtonian (e.g. saline solution) and power-law (e.g. non-Newtonian fluid mud) fluids. A large number of experiments were conducted with different concentrations of both fluid mud mixtures (Kaolinite clay mixed with tap water) and saline solutions in a laboratory tank [dimensions: $4.3m \times 0.25m \times 0.5m$]. In the experiments, different depths of ambient fluid (tap water) were considered. To determine the experimental C_f values for the viscous-buoyancy propagation phase, theoretical analysis was conducted to relate C_f to the experimental measurables. Based upon experimental observations, C_f is shown to relate to *Re* of the gravity currents inversely for both Newtonian and power-law fluids. While Newtonian gravity currents revealed a single value of the constant of proportionality for the C_f–*Re* relationship, power-law gravity currents revealed multiple values of the constant of proportionality that depends on the fluid-mud concentration.

9:57AM G13.00010 ABSTRACT WITHDRAWN –

Monday, November 19, 2012 8:00AM - 10:10AM –

Session G14 Experiments: PIV I 27B - Chair: Barton Smith, Utah State University

8:00AM G14.00001 A super-resolution approach for uncertainty estimation of PIV measurements , BERNHARD WIENEKE, LaVision GmbH, ANDREA SCIACCHITANO, FULVIO SCARANO, TU-Delft — A super-resolution approach is proposed for the a posteriori uncertainty estimation of PIV measurements. The measured velocity field is employed to determine the displacement of individual particle images. A disparity set is built from the residual distance between paired particle images of successive recordings. Within each interrogation window, the disparity set is treated with a statistical analysis to infer the measurement uncertainty: the mean disparity is ascribed to bias errors due to poor particle image sampling or spatial modulation effect; the dispersion of the set is related to precision errors, mainly due to random noise in the recordings and to errors in the PIV interrogation. The performance of the estimator is assessed on a synthetic uniform flow field with varying out-of-plane displacement. The uncertainty is accurately estimated in optimal imaging condition, but underestimated for very small particle images. Experiments are conducted on a water jet experiment, where the actual measurement error is computed as the difference between measured and a reference displacement field estimated from the time redundancy of highly oversampled data. The uncertainty is quantified accurately within 0.1 px.

8:13AM G14.00002 Mean-flow reconstruction by data-assimilation techniques from PIV-measurements of flow over an idealized airfoil , NICOLAS DOVETTA, LadHyX - Ecole Polytechnique, DIMITRY P.G. FOURES, DAMPT - University of Cambridge, DENIS SIPP, ONERA, PETER J. SCHMID, LadHyX - Ecole Polytechnique, BEVERLEY J. MCKEON, GALCIT- CalTech — Measurements (experimental or numerical) typically contain a low-dimensional representation of a high-dimensional flow field. The methodology that extracts the non-measured components of the flow by matching a parametrized model to the data is referred to as data-assimilation. Flow around an idealized airfoil ($Re = 20000$) is measured using time-resolved PIV which produces the mean velocity field by averaging over a sequence of snapshots, as well as the velocity fluctuations. The mean flow is assumed to be known in only part of the flow domain. The assumed relationship between a mean flow measurement and the Reynolds Averaged Navier-Stokes equations is used together with a data-assimilation strategy in order to recover the mean flow everywhere from artificially limited input data. The estimated and measured mean flows are compared to illustrate the potential and effectiveness of the data-assimilation technique.

8:26AM G14.00003 Volumetric PIV with a Plenoptic Camera[1] , BRIAN THUROW, TIM FAHRINGER, Auburn University — Plenoptic cameras have received attention recently due to their ability to computationally refocus an image after it has been acquired. We describe the development of a robust, economical and easy-to-use volumetric PIV technique using a unique plenoptic camera built in our laboratory. The tomographic MART algorithm is used to reconstruct pairs of 3D particle volumes with velocity determined using conventional cross-correlation techniques. 3D/3C velocity measurements (volumetric dimensions of 2.8" x 1.9" x 1.6") of a turbulent boundary layer produced on the wall of a conventional wind tunnel are presented.

[1]This work has been supported by the Air Force Office of Scientific Research,(Grant #FA9550-100100576).

8:39AM G14.00004 Autocorrelation based estimate of particle image density in PIV[1] , SCOTT WARNER, BARTON SMITH, Utah State University, PAVLOS VLACHOS, Virginia Polytechnic Institute and State University — In Particle Image Velocimetry (PIV), the number of particle images per interrogation region, or particle image density, impacts the number of valid vectors and, especially in regions of shear, can impact the uncertainty of PIV. Therefore, any estimate of the uncertainty of PIV requires knowledge of particle image density. An auto-correlation-based method for estimating the local, instantaneous, particle image density is presented. The method is applied to synthetic images to provide an initial estimate of how the autocorrelation peak magnitude varies with known values of particle image density, particle image diameter, illumination intensity, interrogation region size, and background noise. From the synthetic image results, an empirical relationship is developed such that the particle image density is a function of the autocorrelation peak height, particle image diameter, illumination intensity, interrogation region size. Results are also obtained using images from two experimental setups with different seeding particles and flow mediums. The experimental results are compared to manual particle counts and are found to be robust. The effect of varying particle image intensities is also discussed and found to effect the particle image density.

[1]This work was performed with support from Idaho National Laboratory under and LDRD grant monitored by Carl Stoots.

8:52AM G14.00005 PIV Uncertainty Roadmap , BARTON SMITH, Utah State University, STEVEN BERESH, Sandia National Laboratories, PAVLOS VLACHOS, Virginia Tech — The error of PIV measurement has dozens of sources that can broadly be categorized as stemming from the particle motion, particle images, the processing algorithm, post processing, or hardware. Some cause bias errors while others cause random errors. In this talk, we will present our current thinking on the most important of these and how their impact can be assessed. While the theory of PIV is quite advanced, for most PIV uncertainty sources, such an assessment has yet to be performed. Hundreds of studies have focused on the accuracy of PIV, but these studies often cannot be generalized in such a way as to provide a 95% confidence uncertainty band for every instantaneous vector in a PIV data set. Methods for assessment include use of synthetic images, image processing of real images, concomitant measurements, or deviation from best practices. Other assessment may be able to be automated and become part of the vector processing. In order for the PIV community to begin producing meaningful uncertainty estimates for PIV data, it is essential that uncertainty estimation for PIV becomes accessible and affordable.

9:05AM G14.00006 Method to Determine the Minimum Random Uncertainty in PIV Based on Real Images , KYLE JONES, BARTON SMITH, Utah State University — The noise floor, or minimum random uncertainty, of PIV based on the actual acquired PIV images can be determined by generating image pairs with known displacement. Image pairs are acquired with sufficiently small dt such that there is zero displacement between the images. A second image is then shifted by a prescribed amount. By computing a vector field based on these image pairs and calculating the standard deviation of the errors, a random uncertainty can be computed that incorporates the effects of camera noise, particle density, particle images, and displacement. The resultant values are helpful in determining whether the image quality is sufficient to achieve the desired uncertainty. A common PIV experimental setup with seeded water in a glass tank was used. The aperture of the camera lens was varied to achieve a range of particle image diameters. It was found that it is critical to filter the images prior to shifting in order to prevent smearing of the particle images. A Matlab code was written to shift the images by a prescribed, sub-pixel displacement, which were then imported into DaVis and correlated, resulting in displacement vector images. The random and bias errors of the DaVis and PRANA SCC algorithms are also compared for multiple sub-pixel displacements.

9:18AM G14.00007 3D reconstruction and velocity fields of a flame , JONATHON PENDLEBURY, DALE TREE, TADD TRUSCOTT, Brigham Young University — We present three-dimensional internal velocity and shape measurements of an axi-symmetric partially premixed natural gas diffusion flame using Synthetic Aperture particle image velocimetry (SAPIV). This is the first step in demonstrating the technique for fully turbulent premixed flames. It has been shown that there is significant 3-dimensional motion in turbulent flames and knowing the internal fluid structures of flames is vital to understanding the flame properties. For example, fluid strain is directly related to flame extinction and recent measurements are showing strain is related to regions of soot formation and oxidation. Thus, the 3D strain velocity field and strain tensor are needed to both understand flame physics and validate flame theory and models. The current state of the art in flame imaging is Stereoscopic-PIV (2D-3C). Alternatively, SAPIV allows the complete time resolved 3D reconstruction of a measurement volume. The technique refocuses multi-camera viewpoints (8-10) into a refocused image at several depths within a scene, reconstructing a focal stack of images for each time step.

9:31AM G14.00008 Microscopic Light Field Particle Image Velocimetry , TADD TRUSCOTT, BRYCE MCEWEN, Brigham Young University, JESSE BELDEN, Naval Undersea Warfare Center — This work presents the development and analysis of a system that combines the concepts of light field microscopy (Levoy 2006) and particle image velocimetry (PIV) to measure 3D velocities within a micro-volume. Flow at Reynolds numbers in the range of 0.02 to 0.03 was seeded with fluorescent particles and pumped through a micro-channel. The images were post processed to render a stack of 2D refocused images resulting in a 3D focal stack. Subsequently, a multi-pass, 3D PIV algorithm was used to measure channel velocities. Results from PIV analysis were compared with an analytical solution for fully developed cases, and with CFD simulations for developing flows. The relative error and advantages / disadvantages of this system will be presented.

9:44AM G14.00009 Applying Tomographic PIV to Turbulent Taylor-Couette Flows[1] , DANIEL BORRERO-ECHEVERRY, School of Physics and Center for Nonlinear Science, Georgia Institute of Technology, DONALD WEBSTER, School of Civil and Environmental Engineering, Georgia Institute of Technology, MICHAEL SCHATZ, School of Physics and Center for Nonlinear Science, Georgia Institute of Technology — Over the years many techniques have been used to measure velocity fields in Taylor-Couette flows. However, these have been limited to measurements at discrete points (i.e., LDV or hotwire measurements) or planar sections of the flow (i.e., planar or stereo PIV). Tomographic PIV is a strong candidate for extending these measurements to three component volumetric velocity fields. Applying tomographic PIV to Taylor-Couette flows poses some serious challenges (curved interfaces, mechanical vibration, and moving surfaces). We discuss how these issues may be resolved and present temporally and spatially resolved measurements of the structures that form when finite-amplitude perturbations are applied to linearly stable Taylor-Couette flow and trigger the transition to turbulence.

[1]This work is supported by NSF grant CBET-0853691.

9:57AM G14.00010 Propagation of Instantaneously Varying Systematic and Random Uncertainties into the Measurement Mean, Variance, and Covariance , BRANDON M. WILSON, Los Alamos National Laboratory, BARTON L. SMITH, Utah State University — Particle Image Velocimetry (PIV), has numerous error sources, introducing non-linear, instantaneously varying systematic and random errors. Traditional uncertainty quantification (UQ) methods assume linear or constant uncertainty on the time-averaged quantity and may be inadequate for methods with this sort of error behavior. The Uncertainty Surface, and other methods can provide the instantaneous systematic and random errors of a PIV measurement. Equations to propagate these non-linear, instantaneously varying uncertainties into measurement statistics (e.g. mean, variance, and covariance) are derived using the Taylor series uncertainty equations and presented. The mean and variance uncertainty equation validity is verified using Monte-Carlo simulations for various instantaneously varying errors. The effects of the relative magnitude, asymmetry, and variance of random and systematic errors on the measurement uncertainty are demonstrated. The mean and variance uncertainty equations are demonstrated with actual PIV measurements for two experiments. Instantaneous uncertainties are estimated using the Uncertainty Surface method for four error sources: particle image density, diameter, and displacement and gradients. The first experiment consists of a rectangular jet with known error sources. Hot wire measurements are compared to the PIV measurements to assess the accuracy of the mean and variance uncertainties. These uncertainties are also demonstrated for flow through a confined bank of cylinders.

Monday, November 19, 2012 8:00AM - 10:10AM –

Session G15 Biofluids: Large Swimmers III 28A - Chair: David Hu, Georgia Institute of Technology

8:00AM G15.00001 Thrust and power measurements of Olympic swimmers , TIMOTHY WEI, University of Nebraska - Lincoln, VICKI WU, RPI, SEAN HUTCHISON, Ikkos Training, RUSSELL MARK, USA Swimming — Elite level swimming is an extremely precise and even choreographed activity. Swimmers not only know the exact number of strokes necessary to take them across the pool, they also plan to be a precise distance from the wall at the end of their last stroke. Too far away and they lose time by drifting into the wall. Too close and their competitor may slide in before their hand comes forward to touch the wall. In this context, it is important to know, in detail, where and how a swimmer propels her/himself through the water. Over the past decade, state-of-the-art flow and thrust measurement diagnostics have been brought to competitive swimming. But the ability to correlate stroke mechanics to thrust production without somehow constraining the swimmer has here-to-fore not been possible. Using high speed video, a simple approach to mapping the swimmer's speed, thrust and net power output in a time resolved manner has been developed. This methodology has been applied to Megan Jendrick, gold medalist in the 100 individual breast stroke and 4 x 100 medley relay events in 2000 and Ariana Kukors, 2009 world champion and continuing world record holder in the 200 individual medley. Implications for training future elite swimmers will be discussed.

8:13AM G15.00002 Effects of flexibility on bio-inspired aquatic propulsion[1] , PETER DEWEY, BIRGITT BOSCHITSCH, ALEXANDER SMITS, Princeton University — We present the results of an experimental investigation aimed at understanding the role that flexibility plays in bio-inspired aquatic propulsion. A rectangular pitching panel apparatus, where both the flexibility and aspect ratio can be systematically varied, is utilized as a simplified model for bio-inspired propulsion in water at a Reynolds number of 7200. It is found that, when optimized, flexibility can double the thrust produced and propulsive efficiency achieved in comparison to a rigid panel. There is a notable thrust enhancement when the flexible panels are operating near resonance; however, it is found that resonance is not the primary mechanism governing efficient propulsion. Peaks in propulsive efficiency are found below, at, and above the resonant frequency depending on the flexibility of the panel. Finally, a scaling law is derived that is shown to collapse the thrust production, power consumption, and propulsive efficiency data across all panels examined.

[1]The authors would like to acknowledge the generous support from the Office of Naval Research under Program Director Dr. Bob Brizzolara, MURI grant number N00014-08-1-0642.

8:26AM G15.00003 Thrust production of free-to-pivot plates at low Reynolds number , KENNETH GRANLUND, MICHAEL OL, Air Force Research Labs, LUIS BERNAL, University of Michigan — As an abstraction of flapping-wing aerodynamics, rigid flat plates free-to-pivot at the leading edge between incidence angle limits of $\pm 45^\circ$ are considered in rectilinear as well as waving motion in a quiescent fluid. Thrust (lift) and resistive-force are measured, forming a hover Figure-of-Merit (FoM). The evolution of spatial retention of a leading edge vortex is tracked throughout the motion cycle, showing vortex formation shortly after the plate completes its rotation, and in some cases shedding of subsequent vortices after the initial leading edge vortex is ejected. Vortex evolution in rectilinear- vs. rotating and steady vs. accelerating motion is visualized with fluorescent dye illuminated by a laser light sheet at several spanwise stations along the leading edge. Experiments in water reveal a Reynolds number indifference in thrust and FoM for $8{,}000<Re<31{,}000$ based on the maximum velocity of the leading edge. A study on aspect ratio from 3.4 to a nominally 2D-plate also shows an indifference in vortex shedding, force coefficients and FoM. The main operative parameter for aerodynamic coefficients is the stroke-to-chord ratio of the leading edge, with decay in both thrust production and FoM as the ratio approaches unity. Prescribed kinematics of varying phase lead/lag of the pitch- vs. stroke history from free-to-pivot cases shows the effect on attachment of the leading edge vortex and related thrust production and FoM. The effect on flow and force for several orders of magnitude lower Reynolds number are investigated by performing the experiment in varying mixtures of glycerin and water.

8:39AM G15.00004 Swimming Near the Wall[1] , DANIEL QUINN, KEITH MOORED, PETER DEWEY, Princeton University, GEORGE LAUDER, Harvard University, ALEXANDER SMITS, Princeton University — The aerodynamic loads on rectangular panels undergoing heave and pitch oscillations near a solid wall were measured using a 6-axis ATI sensor. Over a range of Strouhal numbers, reduced frequencies and flexibilities, swimming near the wall was found to increase thrust and therefore the self-propelled swimming speed. Experimental particle image velocimetry revealed an asymmetric wake structure with a momentum jet angled away from the wall. Both the thrust amplification and the asymmetric wake structure were verified and investigated further using an in-house inviscid panel method code.

[1]Supported by ONR MURI Grant N00014-08-1-0642

8:52AM G15.00005 On the efficiency of fish like swimming[1] , MICHEL BERGMANN, ANGELO IOLLO, INRIA Bordeaux Sud Ouest / Institut de Mathématiques de Bordeaux, INRIA TEAM MC2 TEAM — The aim of this talk is to present a parametric study of underwater locomotion via numerical simulations. The Navier-Stokes equations are discretized onto a cartesian mesh and the interface between the fluid and the fish is computed using an immersed boundary method. The lagrangian motion of the swimmer is computed from the Newton's laws. We present results showing how the swimming efficiency is influenced by the reynolds number and the swimming law.

[1]This work has been supported by French National Research Agency (ANR) through COSINUS program (project CARPEINTER no. ANR-08-COSI-002).

9:05AM G15.00006 The Effects of Limb Coordination on the Swimming Efficiency of Crayfish , ROBERT GUY, JIAWEI ZHANG, Department of Mathematics, University of California Davis, QINGHAI ZHANG, Department of Mathematics, University of Utah, TIMOTHY LEWIS, Department of Mathematics, University of California Davis — Limbs of crayfish, called swimmerets, move rhythmically in a metachronal wave that progresses from back to front during forward swimming. Neighboring swimmerets maintain phase-lags of about 25% over a wide range of frequencies. This "phase constancy" suggests that there may be mechanical advantages to this stroke pattern. We use the immersed-boundary method to simulate the coupled mechanics of the swimmerets and the surrounding fluid in order to explore how stroke patterns affect swimming efficiency.

9:18AM G15.00007 Model of skin friction enhancement in undulatory swimming , UWE EHRENSTEIN, CHRISTOPHE ELOY, IRPHE, Aix-Marseille Univ, France — To estimate the energetic cost of undulatory swimming, it is crucial to evaluate the drag forces originating from skin friction. This topic has been controversial for decades, some claiming that animals use ingenious mechanisms to reduce the drag and others hypothesizing that the undulatory motion induces a drag increase because of the compression of the boundary layers. In this paper, we examine this latter hypothesis, known as the "Bone–Lighthill boundary-layer thinning hypothesis".[1] Considering a plate of section s moving perpendicular to itself at velocity $U_\perp$ and applying the boundary-layer approximation for the incoming flow, the drag force per unit surface is shown to scale as $\sqrt{U_\perp/s}$. An analogous two-dimensional Navier-Stokes problem by artificially accelerating the flow in a channel of finite height is solved numerically, showing the robustness of the analytical results. Solving the problem for an undulatory plate motion similar to fish swimming, we find a drag enhancement which can be estimated to be of the order of 20 to 100%, depending on the geometry and the motion.

[1]M.J. Lighthill, *Proc. R. Soc. Lond. B* **179**, 125 (1971).

9:31AM G15.00008 Optimal number of waves for ribbon fin propulsion , RAHUL BALE, AMNEET P.S. BHALLA, MALCOLM A. MACIVER, NEELESH A. PATANKAR, Northwestern University — Many species of fish, such as *rajiform, diodontiform, amiiform, gymnotiform* and *balistiform* swimmers, use a ribbon fin as their primary mode of propulsion. It has been observed that each fish species, depending on its size, uses a specific number of waves on its ribbon fin. For example, the black ghost knifefish (10×1cm fin) typically uses 2-2.5 waves while a giant oarfish (3×0.1m fin) uses 6-8 waves. In this work we investigate whether this leads to optimal axial thrust. The axial thrust generated depends on the efficiency with which fin waves transport the fluid backward. We find that there are two competing mechanisms. On the one hand, an increase in wavelength (at fixed amplitude and frequency of the wave), and therefore the wave velocity, leads to an increased ability to transport the fluid backward. This leads to more thrust. On the other hand, longer wavelength leads to shallower waves. This reduces the efficacy to transport the fluid backward and reduces the thrust. The optimal wavelength, and therefore the optimal number of waves, is a result of a balance between the two competing mechanisms. We do our analysis in terms of specific wavelength, which is the wavelength non-dimensionalized by the wave amplitude. We find that the value of the specific wavelength at which the axial thrust is maximized is universal for ribbon fins and it is in agreement with biological data.

9:44AM G15.00009 Shark Skin Bristling: A Passive Flow-Actuated Separation Control Mechanism[1] , AMY LANG, JONATHON SMITH, MICHAEL BRADSHAW, JENNIFER WHEELUS, University of Alabama, PHILIP MOTTA, MARIA HABEGGER, JESSICA DAVIS, University of South Florida, ROBERT HUETER, Mote Marine Laboratory — A collaborative experimental effort between biologists and engineers has proven the separation control capability of shark skin, with a specific focus on the shortfin mako (*Isurus oxyrinchus*) known for its high speed and agility. Biological measurements of the denticles, or scales, as a function of body location (DOI:10.1002/jmor.20047) will be presented together with data on bristling angle of scales and the morphological implications. Results show key regions of high bristling capability to correspond with those most prone to flow separation; these include the tail, flank regions aft of the gills, and on pectoral fins with scale flexibility increasing towards the trailing edge. Fresh shark skin samples were also tested in a water tunnel facility using DPIV and evidence of flow separation control was observed under laminar and tripped boundary layer conditions. It was concluded that the experiments conducted in the Re $\sim 10^5$ range resulted in sufficiently strong backflow induced close to the surface such that the shear threshold to induce bristling on the real skin sample was achieved since flow control at lower Re was not as evident. It is hypothesized that backflow initiated close to the wall in a region of adverse pressure gradient induces localized scale bristling thereby interrupting the subsequent flow development that leads to global flow separation from the surface and increased drag.

[1]Funding from NSF CBET grant 0932352 and US DOD AMRDEC

9:57AM G15.00010 Underlying principles of flexible bio-inspired propulsion: Hydrodynamic wake resonance analysis , KEITH MOORED, PETER DEWEY, ALEXANDER SMITS, Princeton University, HOSSEIN HAJ-HARIRI, University of Virginia — Experiments on flexible pitching panels have demonstrated that flexibility can be utilized to double both thrust production and propulsive efficiency. Yet, the exact mechanisms that lead to efficient locomotion with flexible propulsors are not understood. Using experimental particle image velocimetry data from flexible pitching panels, a linear stability analysis is performed and compared with efficiency data. It is shown that when the driving frequency of motion matches a wake resonant frequency a peak in efficiency occurs not just for rigid panels but also for flexible panels, and that the panel flexibility that leads to global optimally efficient locomotion is the one where a structural resonant frequency is tuned to a wake resonant frequency. The primary mechanism to achieve efficient locomotion is to tune the frequency of motion to a wake resonant frequency, while the secondary mechanism is to tune a structural resonant frequency to a wake resonant frequency. The untuned flexible panels exhibit at most a 46% increase in efficiency over the rigid panel, while the tuned flexible panels attain global optimally efficient locomotion with a 100 to 108% increase in efficiency.

Monday, November 19, 2012 8:00AM - 10:10AM –

Session G16 Biofluids: Aneurysms and Thrombosis 28B - Chair: Shawn Shadden, Illinois Institute of Technology

8:00AM G16.00001 Flow topology in patient-specific abdominal aortic aneurysms during rest and exercise[1] , AMIRHOSSEIN ARZANI, SHAWN SHADDEN, Illinois Institute of Technology — Abdominal aortic aneurysm (AAA) is a permanent, localized widening of the abdominal aorta. Flow in AAA is dominated by recirculation, transitional turbulence and low wall shear stress. Image-based CFD has recently enabled high resolution flow data in patient-specific AAA. This study aims to characterize transport in different AAAs, and understand flow topology changes from rest to exercise, which has been a hypothesized therapy due to potential acute changes in flow. Velocity data in 6 patients with different AAA morphology were obtained using image-based CFD under rest and exercise conditions. Finite-time Lyapunov exponent (FTLE) fields were computed from integration of the velocity data to identify dominant Lagrangian coherent structures. The flow topology was compared between rest and exercise conditions. For all patients, the systolic inflow jet resulted in coherent vortex formation. The evolution of this vortex varied greatly between patients and was a major determinant of transport inside the AAA during diastole. During exercise, previously observed stagnant regions were either replaced with undisturbed flow, regions of uniform high mixing, or persisted relatively unchanged. A mix norm measure provided a quantitative assessment of mixing.

[1]This work was supported by the National Institutes of Health, grant number 5R21HL108272.

8:13AM G16.00002 Virtual Treatment of Basilar Aneurysms Using Shape Memory Polymer Foam , J.M. ORTEGA, Lawrence Livermore National Laboratory, J. HARTMAN, Kaiser Permanente, J.N. RODRIGUEZ, D.J. MAITLAND, Texas A&M University — Numerical simulations are performed on patient-specific basilar aneurysms that are treated with shape memory polymer (SMP) foam. In order to assess the post-treatment hemodynamics, two modeling approaches are employed. In the first, the foam geometry is obtained from a micro-CT scan and the pulsatile blood flow within the foam is simulated for both Newtonian and non-Newtonian viscosity models. In the second, the foam is represented as a porous media continuum, which has permeability properties that are determined by computing the pressure gradient through the foam geometry over a range of flow speeds comparable to those of in vivo conditions. Virtual angiography and additional post-processing demonstrate that the SMP foam significantly reduces the blood flow speed within the treated aneurysms, while eliminating the high-frequency velocity fluctuations that are present prior to treatment. A prediction of the initial locations of thrombus formation throughout the SMP foam is obtained by means of a low fidelity thrombosis model that is based upon the residence time and shear rate of blood. The two modeling approaches capture similar qualitative trends for the initial locations of thrombus within the SMP foam.

8:26AM G16.00003 PIV Measurement of Wall Shear Stress and Flow Structures within an Intracranial Aneurysm Model , RICKY CHOW, EPH SPARROW, U. of Minn., GARY CAMPBELL, Lake Region Medical, AFSHIN DIVANI, U. of Minn., JIAN SHENG, Texas Tech Univ. — The formation and rupture of an intracranial aneurysm (IA) is a debilitating and often lethal event. Geometric features of the aneurysm bulb and upstream artery, such as bulb size, bulb shape, and curvature of the artery, are two groups of factors that define the flow and stresses within an IA. Abnormal flow stresses are related to rupture. This presentation discusses the development of a quasi-3D PIV technique and its application in various glass models at Re = 275 and 550 to experimentally assess at a preliminary level the impact of geometry and flow rate. Some conclusions are to be drawn linking geometry of the flow domain to rupture risk. The extracted results also serve as the baseline case and as a precursor to a companion presentation by the authors discussing the impact of flow diverters, a new class of medical devices. The PIV experiments were performed in a fully index-matched flow facility, allowing for unobstructed observations over complex geometry. A reconstruction and analysis method was devised to obtain 3D mean wall stress distributions and flow fields. The quasi 3D measurements were reconstructed from orthogonal planes encompassing the entire glass model, spaced 0.4mm apart. Wall shear stresses were evaluated from the near-wall flow viscous stresses.

8:39AM G16.00004 CFD-based Thrombotic Risk Assessment in Kawasaki Disease Patients with Coronary Artery Aneurysms[1] , DIBYENDU SENGUPTA, ETHAN KUNG, ANDREW KAHN, JANE BURNS, ALISON MARSDEN, UC San Diego — Coronary aneurysms occur in 25% of untreated Kawasaki Disease (KD) patients and put patients at increased risk for myocardial infarction and sudden death. Clinical guidelines recommend using aneurysm diameter >8mm as the arbitrary criterion for treating with anti-coagulation therapy. This study uses patient-specific modeling to non-invasively determine hemodynamic parameters and quantify thrombotic risk. Anatomic models were constructed from CT angiographic image data from 5 KD aneurysm patients and one normal control. CFD simulations were performed to obtain hemodynamic data including WSS and particle residence times (PRT). Thrombosis was clinically observed in 4/9 aneurysmal coronaries. Thrombosed vessels required twice as many cardiac cycles (mean 8.2 vs. 4.2) for particles to exit, and had lower mean WSS (1.3 compared to 2.8 dynes/cm^2) compared to vessels with non-thrombosed aneurysms of similar max diameter. 1 KD patient in the cohort with acute thrombosis had diameter < 8mm. Regions of low WSS and high PRT predicted by simulations correlated with regions of subsequent thrombus formation. Thrombotic risk stratification for KD aneurysms may be improved by incorporating both hemodynamic and geometric quantities. Current clinical guidelines to assess patient risk based only on aneurysm diameter may be misleading. Further prospective study is warranted to evaluate the utility of patient-specific modeling in risk stratifying KD patients with coronary aneurysms.

[1]NIH R21

8:52AM G16.00005 Mechanisms of thrombolysis acceleration by cavitation , HOPE WEISS[1], PRASHANTH SELVARAJ, UC Berkeley, GOLNAZ AHADI, ARNE VOIE, THILO HOELSCHER, UCSD, KOHEI OKITA, Nihon University, YOICHIRO MATSUMOTO, University of Tokyo, ANDREW SZERI, UC Berkeley — Recent studies, in vitro and in vivo, have shown that High Intensity Focused Ultrasound (HIFU) accelerates thrombolysis, the dissolution of blood clots, for ischemic stroke. Although the mechanisms are not fully understood, cavitation is thought to play an important role in sonothrombolysis. The damage to a blood clot's fibrin fiber network from cavitation in a HIFU field is studied using two independent approaches for an embedded bubble. One method is extended to the more important scenario of a bubble outside a blood clot that collapses asymmetrically creating a jet towards the clot. There is significantly more damage potential from a bubble undergoing cavitation collapse outside the clot compared to a rapidly expanding bubble embedded within the clot structure. Also, the effects of the physical properties of skull bone when a HIFU wave propagates through it are examined by use of computer simulation. The dynamics of a test bubble placed at the focus is used in understanding of the pressure field. All other things being equal, the analysis suggests that skull thickness can alter the wave at the focus, which in turn can change the nature of cavitation bubble dynamics and the amount of energy available for clot damage.

[1]Now at MSOE

9:05AM G16.00006 Evaluation Of Hemolysis Models Using A High Fidelity Blood Model[1] , HUSSEIN EZZELDIN, The George Washington University, MARCO DE TULLIO, Politecnico di Bari, Italy, SANTIAGO SOLARES, University of Maryland, ELIAS BALARAS, The George Washington University — Red blood cell (RBC) hemolysis is a critical concern in the design of heart assisted devices, such as prosthetic heart valves (PHVs). To date a few analytical and numerical models have been proposed to relate either hydrodynamic stresses or RBC strains, resulting from the external hydrodynamic loading, to the expected degree of hemolysis as a function of time. Such models are based on either "lumped" descriptions of fluid stresses or an abstract analytical-numerical representation of the RBC relying on simple geometrical assumptions. We introduce two new approaches based on an existing coarse grained (CG) RBC structural model, which is utilized to explore the physics underlying each hemolysis model whereby applying a set of devised computational experiments. Then, all the models are subjected to pathlines calculated for a realistic PHVs to predict the level of RBC trauma. Our results highlight the strengths and weaknesses of each approach and identify the key gaps that should be addressed in the development of new models. Finally, a two-layer CG model, coupling the spectrin network and the lipid bilayer, which provides invaluable information pertaining to RBC local strains and hence hemolysis.

[1]We acknowledge the support of NSF OCI-0904920 and CMMI-0841840 grants. Computing time was provided by XSEDE

9:18AM G16.00007 A non-discrete method for residence time calculation as an indicator of thrombus formation in cardiovascular applications , MAHDI ESMAILY MOGHADAM, ALISON MARSDEN, University of California San Diego — Cardiovascular simulations provide a promising means to predict risk of thrombosis in grafts, devices, and surgical anatomies in adult and pediatric patients. Although the pathways for platelet activation and clot formation are not fully understood, recent findings suggest that thrombosis risk correlates with the presence of recirculation regions with high residence time (RT). Current approaches for calculating RT are often based on releasing a finite number of Lagrangian particles in the flow and calculating RT by tracking their pathways. However, this method requires several simulations for a single case study, each of which requires releasing a significant number of particles, to obtain temporal and spatial convergence. In this work, we introduce a new non-discrete method, in which RT is calculated in an Eulerian non-discrete framework, using the advection-diffusion equation. Starting with an existing and a newly developed intuitive definition for the RT, the formulation for calculating RT in a region of interest is presented. The physical significance and sensitivity of each measure of RT is discussed and an extension of these definitions to a point-wise value is presented. Application to simulations of shunt insertion for single ventricle heart patients is demonstrated.

9:31AM G16.00008 Computational Study of Non-Physiological Hemodynamics in the Cephalic Arch[1] , KEVIN CASSEL, MICHAEL BOGHOSIAN, S.M. JAVID MAHMOUDZADEH, Illinois Institute of Technology, MARY HAMMES, University of Chicago Medical Center — Numerical simulations of the unsteady, two-dimensional, incompressible Navier-Stokes equations are performed for the flow in a two-dimensional geometry created from radiological images and Doppler flow measurements of the cephalic arch in dialysis patients with a brachiocephalic fistula (surgically placed direct arterial-venous connection). The simulations are performed before insertion of the fistula and at subsequent time intervals as the cephalic vein arterializes over a period of three to six months. A mature fistula, with increased diameter and flow rate, can exhibit Reynolds numbers that are more than one order of magnitude larger than that of the pre-fistula vein. We evaluate the effect of this increased (physiologically abnormal) Reynolds number on flow structures and wall shear stresses through the curved cephalic arch, which is a site prone to stenosis in fistula patients. The long-term goal is to investigate if the development of initimal hyperplasia and stenoses correlates with wall shear stresses or other hemodynamic variables obtained using computational hemodynamics.

[1]Research supported by the National Institute of Diabetes And Digestive And Kidney Diseases of the National Institutes of Health under Award Number R01DK090769.

9:44AM G16.00009 Quantifying Turbulent Kinetic Energy in an Aortic Coarctation with Large Eddy Simulation and Magnetic Resonance Imaging , JONAS LANTZ, Department of Management and Engineering, Linkoping University, TINO EBBERS, Department of Medical and Health Sciences, Linkoping University, MATTS KARLSSON, Department of Management and Engineering, Linkoping University — In this study, turbulent kinetic energy (TKE) in an aortic coarctation was studied using both a numerical technique (large eddy simulation, LES) and in vivo measurements using magnetic resonance imaging (MRI). High levels of TKE are undesirable, as kinetic energy is extracted from the mean flow to feed the turbulent fluctuations. The patient underwent surgery to widen the coarctation, and the flow before and after surgery was computed and compared to MRI measurements. The resolution of the MRI was about 7x7 voxels in axial cross-section while 50x50 mesh cells with increased resolution near the walls was used in the LES simulation. In general, the numerical simulations and MRI measurements showed that the aortic arch had no or very low levels of TKE, while elevated values were found downstream the coarctation. It was also found that TKE levels after surgery were lowered, indicating that the diameter of the constriction was increased enough to decrease turbulence effects. In conclusion, both the numerical simulation and MRI measurements gave very similar results, thereby validating the simulations and suggesting that MRI measured TKE can be used as an initial estimation in clinical practice, while LES results can be used for detailed quantification and further research of aortic flows.

9:57AM G16.00010 Multiscale Simulation for the Initial Stage of Thrombus Formation , SHU TAKAGI, The University of Tokyo, SATOSHI II, Osaka University, SEIJI SHIOZAKI, RIKEN, NORIO SHIMAMOTO, KAZUYASU SUGIYAMA, YOICHIRO MATSUMOTO, The University of Tokyo — Thrombosis is regarded as one of the most important diseases, which cause the myocardial and cerebral infarctions. The thrombus formation is strongly related to the multiscale coupling phenomena from molecular scale protein-protein interaction to continuum scale in blood flow. Initially, platelets start aggregate at the injured vessel wall, where von Willebrand Factor (vWF) is attached. The Glycoprotein, GPIb-α, on platelet membrane starts showing ligand-receptor interaction with this vWF and platelets start aggregating around this spot. In the present study, multiscale coupling method is developed to simulate the initial stage of thrombus formation. In this method, the molecular scale interactions between vWF and GPIb-α is solve using the stochastic Monte Carlo simulations and the binding force at each computational cell is calculated. Then the force is directly coupled with the continuum scale simulation using finite different method. The results illustrate that fluctuation given by the motion of red blood cells plays an important role for the platelets to adhere to the injured vessel walls.

Monday, November 19, 2012 8:00AM - 10:10AM –

Session G17 Biofluids: Microswimmers Experiments I 28C - Chair: Sung Kwon Cho, University of Pittsburgh

8:00AM G17.00001 Experiments and models of low Reynolds number flows generated by a precessing rod over a plane[1] , JAMES MARTINDALE, ROBERTO CAMASSA, RICHARD M. MCLAUGHLIN, LEANDRA VICCI, University of North Carolina, LONGHUA ZHAO, University of Minnesota, UNC NSF RTG FLUIDS TEAM COLLABORATION — Slender body asymptotics and experiments are developed to emulate dynamics biological interest such as primary cilia in developing embryos. Experiments are performed using high viscosity silicon oil with magnetically actuated precessing rod in a table-top setup. Stereoscopic Lagrangian tracking show quantified long-time agreement with an appropriately imaged slender body theory to enforce the no-slip boundary condition of the floor. In contrast, breaking symmetry by a bent rod creates additional flow components which destroy quantitative short time agreement with the theory while maintaining its qualitative features including the creation of large scale Lagrangian tori. Higher order asymptotic corrections are implemented and compared in an attempt to restore quantitative predictability. Direct comparison with 3D stereoscopic PIV measurements will be presented.

[1]NSF RTG DMS-0502266, NSF RTG DMS-0943851, and NSF DMS-1009750

8:13AM G17.00002 Microscale underwater propulsion by oscillating air bubble columns[1] , JIAN FENG, SUNG KWON CHO, University of Pittsburgh — We will present microscale underwater propulsion using oscillating air bubble columns. A cylindrical air bubble column is formed in a microchannel immersed in water (one end is open and directly exposed to water and the other is closed). Under an external acoustic field, the cylindrical air bubble column axially oscillates and generates a net flow near the open end of the microchannel. This net flow can generate a propelling force depending on many parameters such as oscillating frequency and amplitude. In this presentation, we will report and discuss detailed fabrication, testing and experimental results of this micvroscale propulsion.

[1]This work is supported by NSF (ECCS-1029318).

8:26AM G17.00003 Performance of three different artificial swimmers in Newtonian and complex fluids , F. GODINEZ, R. ZENIT, Universidad Nacional Autonoma de Mexico, E. LAUGA, University of California at San Diego — We present an experimental investigation of three simple swimming devices at low Reynolds number. Each swimmer is composed of a magnetic head attached to a propulsive tail. The robots are driven by an external magnetic field and three different kinds of tails are used: (i) a flexible filament periodically oscillated (the flexible oar mechanism); (ii) a rigid helical filament rotated by the external field (the corkscrew mechanism); (iii) a flexible filament that, when rotated by the field, acquires a conical helical shape (a hybrid case). Each swimmer is tested in two different fluids with the same shear viscosity, a Newtonian and a Boger fluid. Surprisingly, even though the tests were conducted with the same fluid, the results for the viscoelastic fluid are contrastingly different. The device based on flexible oar mechanism swims faster in the Boger fluid than in the Newtonian one; on the contrary, the hybrid device swims at lower speeds in the Boger fluid than in the Newtonian one. And unexpectedly, the device based on the corkscrew mechanism practically swims at the same velocity in both fluids. These results, suggest that the swimming performance of a biomimetic device strongly depends on the details of the swimming actuation. We can conclude that a general viscoelastic effect

8:39AM G17.00004 Swimming of *Chlamydomonas reinhardtii* in weakly elastic fluids[1] , JING YANG, JERRY GOLLUB, Department of Physics, Haverford College, PAULO ARRATIA, University of Pennsylvania — The swimming behavior of the algae *Chlamydomonas reinhardtii* in weakly elastic fluids is investigated in experiments using microscopy and tracking methods. The effects of fluid viscosity and elasticity on the swimming speed, flagellar shape, beating frequency, and efficiency are examined. Here, the fluid viscosity is varied using water and sucrose solutions, while fluid elasticity is introduced by adding flexible polymer CMC (carboxymethyl cellulose) to the buffer solution. Swimming experiments are performed in a thin-film apparatus equipped with a microscope and high-speed camera. We find that even small amounts of fluid elasticity can have a significant effect on the swimming kinematics and dynamics of *Chlamydomonas* because of the relatively high beating frequency of its flagella (50-60 Hz). For example, the *Chlamydomonas* swimming speed is hindered by fluid elasticity compared to Newtonian fluids. In addition, the algae swimming speed decreases as the fluid elasticity is increased.

[1]This research is supported by the NSF through grant DMR-1104705.

8:52AM G17.00005 Tracing the run-flip motion of an individual bacterium , BIN LIU, School of Engineering, Brown University, MICHAEL MORSE, JAY TANG, Department of Physics, Brown University, THOMAS POWERS, KENNETH S. BREUER, School of Engineering, Brown University — We have developed a digital 3D tracking microscope in which the microscope stage follows the motion of an individual motile microorganism so that the target remains focused at the center of the view-field. The tracking mechanism is achieved by a high-speed feedback control through real-time image analysis and the trace of the microorganism is recorded with submicron accuracy. We apply this tracking microscope to a study of the motion of an individual Caulobacter crescentus, a bacterium that moves up to 100 microns (or 50 body lengths) per second and reverses its direction of motion occasionally by switching the rotation direction of its single helical flagellum. By tracking the motion of a single cell over many seconds, we show how a flip event occurs with submicron resolution and how the speed of a single cell varies over time and with the rotational rate of the flagellum. We also present statistics for the run-reverse dynamics of an ensemble of cells.

9:05AM G17.00006 Characterization of gyrotactic swimmers using digital holographic microscopy , MICHAEL BARRY, MIT, Dept. of Mechanical Engineering, WILLIAM M. DURHAM, University of Oxford, Dept. of Zoology, ANWAR CHENGALA, University of Minnesota, Dept. of Civil Engineering, JIAN SHENG, Texas Technical University, Dept. of Mechanical Engineering, ROMAN STOCKER, MIT, Dept. of Civil and Environmental Engineering — Observations from the ocean reveal that motility can exert a strong influence on the spatial distribution of phytoplankton at multiple scales, from thin layers and harmful algal blooms to Kolmogorov-scale accumulations in turbulence. Aside from a few model organisms, however, little is known about the fundamental motility characteristics of marine phytoplankton species, in particular their stability and the noise in their swimming orientation. In the absence of fluid flow, a phytoplankter's swimming direction is governed by the competition between an intrinsic stabilizing torque and stochastic fluctuations resulting from noise in the flagellar beat. These two processes can be parameterized by a gyrotactic reorientation time scale and an effective rotational diffusivity, respectively. Here we obtain measurements of these two parameters by using digital holographic microscopy to capture three-dimensional trajectories of phytoplankton. Novel inverse techniques are applied to individual tracks, which are analyzed for noise in the swimming direction and the rate of reorientation to the vertical. This approach can easily be extended to other species, promising to improve our understanding of how the interaction of motility and flow affects the distribution of phytoplankton communities.

9:18AM G17.00007 ABSTRACT MOVED TO E17.00006 –

9:31AM G17.00008 Experimental Study on the Euglena gracilis for Micro-Transportation using a Phototatic Control[1] , JIHOON KIM, Sungkyunkwan University, VU DAT NGUYEN, Enjet, Inc., DOYOUNG BYUN, Sungkyunkwan University — Recently, there has been growing interests in micro or nano-scale biological organisms for the micro-robotics to develop actively controlled micro or nano-level machines. The Euglena gracilis is a genus of unicellular protists, whose body size ranges from 30 to 70 μm. The Euglena gracilis contains an eyespot, a primitive organelle that filters sunlight into the light-detecting, photo-sensitive structures. It actively swims at the base of the flagellum. In this study, we investigated the controllability of Euglena gracilis for transporting a structure attaching itself. When a LED light is detected, the Euglena gracilis accordingly adjust its position to enhance photosynthesis. Using the phototactic control, we achieved the efficient transportation of a micro-structure.

[1]Partially funded by the Basic Science Research Program through the National Research Foundation of Korea(NRF, 2011-0016461) and the Industrial Core Technology Development Project through the Ministry of Knowledge and Commerce.

9:44AM G17.00009 The effect of shear thinning viscosity on the performance of low Reynolds number swimmers , R. ZENIT, F. GODINEZ, Universidad Nacional Autonoma de Mexico, C. BELLEVILLE, ENSEEIHT, France, E. LAUGA, University of California at San Diego — In addidion to viscoelastic effects, biological fluids can also show shear-thinning viscosity as part of their non Newtonian behavior. To assess the effect of a varying viscosity with shear rate on the performance of swimming, we conducted experiments using two types of magnetically driven swimmers. We consider oscillating flexible tail and rotating rigid coil devices to test this effect. We prepared carbopol-based inelastic shear-thinning fluids with different values of the thinning coefficient, n, and an equivalent Newtonian liquid for comparisons. The motion was filmed and the swimming velocity was measured via digital image processing. We found that the swimming efficiency changes in an important manner if the fluid does not have constant viscosity. We will present and discuss our preliminary results. To our knowledge, this effect has not been addressed in the specialized literature to date.

9:57AM G17.00010 Flow visualization study on the near-surface motility of a flagellar propeller[1] , DONGWOOK YIM, Seoul National University, JAEHYEONG CHO, SONGWAN JIN, Korea Polytechnic University, JUNG YUL YOO, Seoul National University — Understanding of the near-surface motility of microorganisms is important in many bioengineering applications including the initial formation of biofilms and energy-efficient propulsion system which is the most important part of microrobots. In particular, a new type of propeller that is optimized for low Reynolds numbers is required to propel a small object in a medium where the flow is dominated by viscous force rather than inertial force. A propeller in the shape of a bacterial flagellum seems an appropriate choice for this purpose. Thus, in this study, we carried out a flow visualization study on the velocity field near the solid surface, induced by a spring-like propeller inspired by the *E. coli* flagellum, by using a macroscopic model and applying stereoscopic particle image velocimetry. Silicone oil, which has a kinematic viscosity 100,000 times that of water, was used as the working fluid to generate the low Reynolds number condition for the macroscopic model. Thrust, torque, and velocity were measured as functions of pitch and rotational speed, and the efficiency of the propeller was calculated from the measured results.

[1]The supports of the Basic Science Research Program (2009-0071117) and also by the Priority Research Centers Program (2011-0029613) through the NRF funded by the MEST, Republic of Korea are gratefully acknowledged

Monday, November 19, 2012 8:00AM - 10:10AM –

Session G18 Biofluids: Speech and Vocal 28D - Chair: Michael Krane, Penn State University

8:00AM G18.00001 Aeroacoustic measurements in a human airway model[1] , MICHAEL MCPHAIL, ELIZABETH CAMPO, MICHAEL KRANE, Penn State University — Flow and acoustic measurements are presented for a vocal tract-like geometry with a rigid constriction as a prelude to a study of a compliant constriction that models the vocal folds. Optical flow measurements were taken at the inlet of the constriction and downstream in the jet region. Pressure and acoustic measures were taken on either side of the constriction. Volume flow, two-dimensional flow fields, and radiated sound will be presented for a range of driving pressures. Measurements are used to assess the resistance of the constriction and the measures of the aeroacoustic source. The measurements serve as a validation case for computational aeroacoustic simulations.

[1]Acknowledge support from NIH and PSU-ARL E&F program.

8:13AM G18.00002 Aeroacoustic behavior of vocal fold models from acoustic measurements[1] , MICHAEL KRANE, ELIZABETH CAMPO, MICHAEL MCPHAIL, Penn State University — Measurements of the sound field of the Penn State Human Airway Model (HAM) are used to deduce the aeroacoustic behavior of vibrating vocal fold models. In particular, the distinctions between reflection, transmission and source behavior are sought as a means to quantify source-filter interaction. The acoustic measurements are conducted using 5 microphones located along the airway model axis. Phase-corrected signals from these microphones are used to compute the right-and left-running wave components on either side of the model vocal folds. In combination with theory, these cross-spectra are used to estimate the frequency dependence of the vocal fold reflection and transmission coefficients, as well as the aeroacoustic "voice" monopole and dipole source spectra.

[1]Acknowledge support from NIH and PSU-ARL E&F program.

8:26AM G18.00003 Empirical estimates of aeroacoustic source behavior for vocal fold models having male and female geometry[1] , ELIZABETH CAMPO, MICHAEL MCPHAIL, MICHAEL KRANE, Penn State University — Measurements in the Penn State Human Airway Model (HAM) are used to estimate the aeroacoustic source spectra for vocal fold models models built to mimic the behavior of adult male and female humans. A central unanswered question in voice production is how to reliably predict how a change in physiology results in a change in the sound of the voice. Even differences such as those between normal adult males and females are still not fully explained. A combination of trangslottal pressure, radiated sound, volume flow and high speed imaging measurements in the HAM are presented. The theoretical basis for the source estimates is presented to show how the measurements lead to source strength estimates. This study is a first step in establishing how modifications in vocal fold geometry affect the voice's aeroacoustic source strength.

[1]Acknowledge support from NIH and PSU-ARL E&F program.

8:39AM G18.00004 Vocal Folds Simulations with Contact Algorithm , JUBIAO YANG, LUCY ZHANG, Rensselaer Polytechnic Institute — Our modified IFEM numerical algorithm was able to simulate vocal folds vibration, and demonstrates relations among vibration frequencies and several fluid and solid parameters successfully. However, although interactions between the solid, i.e. the vocal folds, and the fluid, i.e. air, are well handled, contacts and forces between solid parts, namely vocal folds, have been neglected based on the assumption that their influence is very limited, which may not hold true. To more accurately predict motion and deformation of the vocal folds, and to evaluate its effects on airflow, a contact algorithm is implemented to model when the glottis is completely shut off. The algorithm is developed to decide whether solid parts have made contact, to calculate the interaction force in between, and to decide when they again set apart. This contact algorithm correctly models the interaction between the vocal folds instead of neglecting the unrealistic overlapping of two solid parts, and shows the deformation of vocal folds caused both by the airflow and by the impact with each other. Our result capture the accurate vocal folds behaviors when two vocal folds are approaching each other in full-space cases, and when a vocal fold is approaching the symmetry-line wall in half-space cases.

8:52AM G18.00005 A coupled experimental-numerical framework for fluid-structure interaction studies: towards a pseudo-self-oscillating vocal fold facility , DAVID SOMMER, SEAN D. PETERSON, University of Waterloo — Voiced speech is a complex process that involves coupled interactions between expelled air and the vocal fold structure. Numerical simulations of this process are difficult due to the unsteady nature of the flow and boundary conditions, while experimental investigations are generally limited in the structural modeling. To bridge this gap, an experimental platform is investigated that couples a mechanical flow facility featuring instrumented and actuated walls, with a numerical structure solver. Specifically, a proof-of-concept experimental apparatus consisting of a flat plate oriented normal to a uniform jet is developed. The plate is instrumented with pressure sensors, which record the pressure distribution caused by the impinging jet. A real-time controller reads the pressure distribution and computes the integrated force on the plate. The resulting force is applied to a numerical structure model comprising a spring-mass-damper system, in which the dynamical parameters can be adjusted in software. The axial position and velocity of the plate are updated in real time based upon the numerical dynamics solution. In the future, this experimental facility will be extended to model two degrees of freedom asymmetric vocal fold motion with full fluid coupling. Pressure sensors distributed across the solid interface, as opposed to direct force sensors, will help explicate the effect of fluid-structure coupling on tissue loading and flow properties, thus allowing for more detailed validation and improvement of computational models.

9:05AM G18.00006 Three-Dimensional Flow Separation Induced by a Model Vocal Fold Polyp[1] , KELLEY C. STEWART, BYRON D. ERATH[2], MICHAEL W. PLESNIAK, The George Washington University — The fluid-structure energy exchange process for normal speech has been studied extensively, but it is not well understood for pathological conditions. Polyps and nodules, which are geometric abnormalities that form on the medial surface of the vocal folds, can disrupt vocal fold dynamics and thus can have devastating consequences on a patient's ability to communicate. A recent in-vitro investigation of a model polyp in a driven vocal fold apparatus demonstrated that such a geometric abnormality considerably disrupts the glottal jet behavior and that this flow field adjustment was a likely reason for the severe degradation of the vocal quality in patients. Understanding of the formation and propagation of vortical structures from a geometric protuberance, and their subsequent impact on the aerodynamic loadings that drive vocal fold dynamic, is a critical component in advancing the treatment of this pathological condition. The present investigation concerns the three-dimensional flow separation induced by a wall-mounted prolate hemispheroid with a 2:1 aspect ratio in cross flow, i.e. a model vocal fold polyp. Unsteady three-dimensional flow separation and its impact of the wall pressure loading are examined using skin friction line visualization and wall pressure measurements.

[1]Supported by the National Science Foundation, Grant No. CBET-1236351 and GW Center for Biomimetics and Bioinspired Engineering (COBRE).
[2]currently at Clarkson University

9:18AM G18.00007 Resolving pressure from DPIV measurements in a dynamically scaled-up vocal fold model[1] , LORI LAMBERT, University of Nebraska - Lincoln, MICHAEL KRANE, Penn State - Applied Research Lab, ERICA SHERMAN, TIMOTHY WEI, University of Nebraska - Lincoln — This presentation highlights application of control volume analysis to DPIV measurements in a dynamically scaled human vocal fold model. For the first time spatially and temporally resolved pressure field information can be extracted from voice experiments. The vocal fold model was built around a computer driven mechanism that replicates both the transverse vibrations as well as the streamwise rocking of human vocal folds. A range of experiments were conducted corresponding to 50 – 200 Hz life frequencies. Volumetric flow rate and maximum velocity measurements will be presented for several control surfaces in the glottal flow. The direction of the glottal jet (coandă) was noted for each instantaneous oscillation cycle. The pressure forces acting in the glottis were calculated using the streamwise and transverse linear momentum equations and control volume analysis. In addition to serving as a baseline study, data from these experiments provide the comparison for follow on studies of diseased and abnormal vocal fold vibrations.

[1]Supported by NIH.

9:31AM G18.00008 Natural and forced asymmetries in flow through a vocal fold model[1] , BETHANY DRAIN, LORI LAMBERT, University of Nebraska - Lincoln, MICHAEL KRANE, Penn State - Applied Research Lab, TIMOTHY WEI, University of Nebraska - Lincoln — Much of the complexity and richness of voice production stems from asymmetries in flow through the vocal folds. There are naturally occurring asymmetries, such as the Coanda effect ($i.e.$ deviation of the glottal jet from the centerline as air passes through the nominally symmetric vocal folds). There are also asymmetries which arise from disease or dysfunction of the vocal folds. This study uses DPIV measurements in a dynamically scaled-up human vocal fold model to compare the flow characteristics between symmetric versus asymmetric oscillations. For this study, asymmetries were introduced by running one vocal fold out of phase with the other. Three phase lags, 0°, 18° and 36°, were examined over a range of frequencies corresponding to the physiological frequencies of 50-200 Hz. Control volume analysis was applied and time traces of terms from the conservation of linear momentum equation were generated. This allowed analysis of how differences in the glottal jet flow manifest themselves in the fluid pressure field. In addition, further examination of the Coanda effect in the context of fluid pressure will be discussed.

[1]Supported by NIH.

9:44AM G18.00009 In Vitro Microfluidic Models of Mucus-Like Obstructions in Small Airways , MOLLY K. MULLIGAN, Technion - Israel Institute of Technology, JAMES B. GROTBERG, University of Michigan, JOSUÉ SZNITMAN, Technion - Israel Institute of Technology — Liquid plugs can form in the lungs as a result of a host of different diseases, including cystic fibrosis and chronic obstructive pulmonary disease. The existence of such fluid obstructions have been found as far down in the bronchiole tree as the sixteenth generation, where bronchiole openings have diameters on the order of a hundred to a few hundred microns. Understanding the propagation of liquid plugs within the bifurcating branches of bronchiole airways is important because their presence in the lungs, and their rupture and break-up, can cause injury to the epithelial cells lining the airway walls as a result of high wall shear stresses. In particular, liquid plug rupture and break-up frequently occurs at airway bifurcations. Until present, however, experimental studies of liquid plugs have generally been restricted to Newtonian fluids that do not reflect the actual pseudoplastic properties of lung mucus. The present work attempts to uncover the propagation, rupture and break-up of mucus-like liquid plugs in the lower generations of the airway tree using microfluidic models. Our approach allows the dynamics of mucus-like plug break-up to be studied in real-time, in a one-to-one in vitro model, as a function of mucus rheology and bronchial tree geometry.

9:57AM G18.00010 Yield Stress Effects on Mucus Plug Rupture[1] , YINGYING HU, SHIYAO BIAN, Department of Biomedical Engineering, University of Michigan, JOHN C. GROTBERG, University of Illinois at Chicago, SHUICHI TAKAYAMA, JAMES B. GROTBERG, Department of Biomedical Engineering, University of Michigan — Mucus plugs can obstruct airways, resulting in lost gas exchange and inflammation. Yield stress, one of the significant rheological properties of mucus, plays a significant role in plug rupture. We use carbopol 940 gels as mucus simulants to study dynamics of mucus plug rupture in experiments. Yield stress increases with gel concentration increasing (0.1%$\sim$0.3%). The yield stress of the 0.2% gel is about 530 dyn/cm^2, which can simulate normal mucus. A 2D PDMS channel is used to simulate a collapsed airway of the 12th generation in a human lung. Plug rupture is driven by a pressure drop of 1.6×10^4 $\sim$$2.0\times10^4$ dyn/cm^2. Initial plug length varies from half to two times the half channel width. A micro-PIV technique is used to acquire velocity fields during rupture, from which wall shear stress is derived. Plug shortening velocity increases with the pressure drop, but decreases with yield stress or the initial plug length. Wall shear stress increases with yield stress, which indicates more potential damage may occur to epithelial cells when pathologic mucus has a high yield stress. Near the rupture moment, a wall shear stress peak appears at the front of the film deposited by the plug during rupture.

[1]This work is supported by NIH: HL84370 and HL85156.

Monday, November 19, 2012 8:00AM - 10:10AM –

Session G19 Surface Tension II 28E - Chair: Karen Daniels, North Carolina State University

8:00AM G19.00001 Surfactant-driven fracture of gels: Initiation , JOSHUA BOSTWICK, MARK SCHILLACI, KAREN DANIELS, NC State University — A droplet of surfactant spreading on a gel substrate can produce fractures on the gel surface, which originate at the contact-line and propagate outwards in a star-burst pattern. Experiments show that the number of arms is controlled by the ratio of surface tension contrast to the gel's shear modulus. To further understand the mechanism behind crack initiation, we model the gel as a linear elastic solid and compute the state of stress that develops within the substrate from the uncompensated contact-line forces. The elastic solution yields an effective metric to predict the number of fractures. We also show that the depth of the gel is critical parameter in the fracture process, as it can help mitigate large surface tractions. This observation is confirmed in experiments.

8:13AM G19.00002 Surfactant-driven fracture of gels: Growth , KAREN DANIELS, MARK SCHILLACI, JOSHUA BOSTWICK, NC State University — A droplet of surfactant spreading on a gel substrate can produce fractures on the gel surface, which originate at the contact-line and propagate outwards in a star-burst pattern. Fractures have previously been observed to initiate through a thermal process, with the number of fractures controlled by the ratio of surface tension differential to gel shear modulus. After the onset of fracture, experiments show the arm length grows with universal power law $L = t^{3/4}$ that does not scale with any material parameters (Daniels et al. 2007, PRL), including super-spreading surfactants (Spandangos et al. 2012, Langmuir). We develop a model for crack growth controlled by the transport of an inviscid fluid into the fracture tip. While treating the gel as a linear material correctly predicts power-law growth, we find that only by considering a Neo-Hookean (incompressible) material do we obtain agreement with the experiments.

8:26AM G19.00003 Hole-Closing of a Surfactant Layer on a Thin Fluid Film , RACHEL LEVY, MATTHEW HIN, M. RICHARD SAYANAGI, ERIC AUTRY, Harvey Mudd College, JEFFREY WONG, UCLA, KAREN DANIELS, NC State University — The spreading of surfactants on a thin fluid layer has been most commonly studied in an outward-spreading geometry. We perform simulations and experiments on the inverse, the inward spreading of surfactant into a clean disk-shaped region, known as hole-closing. In both cases, we observe that the inward force from the surface tension gradient produces a transient distention, in which the underlying fluid is raised within the closing region. We observe that the height of the distension is controlled by a combination of fluid depth and the surface tension gradient between the two regions. We compare the evolution of the distension height over time to a coupled system of partial differential equations that have been used to model surfactant spreading for more than two decades.

8:39AM G19.00004 The Adventures of the Diving-Bell Spider , RAPHAELE THEVENIN, GUILLAUME DUPEUX, KEYVAN PIROIRD, CHRISTOPHE CLANET, DAVID QUERE, Physique et Mecanique des Milieux Heterogenes, CNRS, ESPCI, Paris France - Ladhyx, CNRS, Ecole Polytechnique, Palaiseau, France, INTERFACES & CO. TEAM — The Argyroneta Aquatica is a unique spider that has every features of a usual terrestrial spider, but constantly lives under water. To however still be able to breath oxygen, it builds an underwater bell of air (hence its other name "the diving-bell spider"): using its superhydrophobic abdomen, it pulls an air bubble at the surface by leaving the latter very rapidly. It then enters the bell formed under aquatic plants or under its under-water web, and leaves it more slowly so as to entrain the least air possible. We study these dynamics that take place at the air/water interfaces. We reduce the spider to two beads, one for the hydrophobic abdomen, one for the hydrophilic head, and measure and model the air entrainment according to the size and surface properties of the abdomen and to the velocity of motion.

8:52AM G19.00005 Tension induced phase transitions in biomimetic fluid membranes , MARC SHAPIRO, PETIA VLAHOVSKA, Brown University — Membranes in eukaryotic cells are mixtures of hundreds of lipid species. The lipid diversity enables membranes to phase separate and form domains, called rafts, which play a critical role in cell functions such as signaling and trafficking. The phase transitions underlying raft formation have been extensively studied as a function of temperature and composition. However, the third dimension of the phase diagram, i.e., the tension (2D pressure), is still unexplored because membrane tension is difficult to control and quantify. To overcome this challenge, we develop two approaches, capillary micromechanics and electrodeformation, in which the tension is regulated by the area dilation accompanying deformation of a vesicle (a closed membrane). The first technique consists of forcing an initially quasi-spherical vesicle through a tapered glass microcapillary, while the second method utilizes uniform electric fields to deform the vesicle into an ellipsoid. Domains are visualized using a fluorescent dye, which preferentially partitions in one of the phases. The experimental results suggest that the miscibility temperature (at which domains form in an initially homogeneous membrane) increases with applied tension. Domain motions and coarsening are also investigated.

9:05AM G19.00006 Wicking flow in optimized capillary channels , BRUNO FIGLIUZZI, CULLEN BUIE, Massachusetts Institute of Technology — Many technological applications rely on the phenomenon of wicking flow induced by capillarity. However, despite a continuous interest on the subject, the influence of the geometry of the capillary channel on the dynamics of wicking is still poorly understood. In the case of a cylinder, the well-established Washburn law indicates that, at short time, the height of the rising liquid increases as the square root of time. However, Reyssat et al. demonstrated that shape variations affect the dynamics of the capillary rise at longer times. In numerous applications, being able to favor wicking in a capillary channel is a key issue. Starting from the Washburn-Lucas equation, we've developed a model describing the capillary rise of a liquid in a tube of varying circular cross-section. In this model, the dynamics of wicking is described by an ordinary differential equation whose second term depends on the shape of the capillary channel. Using optimal control theory, we have designed optimal shapes which favor wicking flow. Numerical simulations were conducted which show that the height of the rising liquid is up to 40 percent higher with the optimized shapes than with a cylinder tube of optimal radius.

9:18AM G19.00007 Capillary rise within superhydrophilic channel , JUNGCHUL KIM, Seoul National University, MYOUNG-WOON MOON, Korea Institute of Science and Technology, L. MAHADEVAN, Harvard University, HO-YOUNG KIM, Seoul National University — While the capillary rise within smooth channels is a classical topic in hydrodynamics, the dynamics of liquid rise through superhydrophilic, microscopically rough channels have rarely been studied so far. Here we experimentally show that within superhydrophilic channels, a bulk flow rises against gravity in a similar fashion to one in smooth channels in the initial stages. However, as the bulk approaches Jurin's height, a thin film that wicks into the rough surface emerges, a novel feature characteristic to superhydrophilic capillary rise. We construct a scaling law to explain the wicking rate of the thin film, which depends on the bulk height as well as the surface roughness and liquid properties. This study is potentially useful in understanding transport of sap through porous xylems of plants.

9:31AM G19.00008 ABSTRACT WITHDRAWN –

9:44AM G19.00009 Capillary interactions between spherical Janus particles at liquid-liquid interfaces , HOSSEIN REZVANTALAB, SHAHAB SHOJAEI-ZADEH, Rutgers, The State University of New Jersey — We study the non-equilibrium behavior of Janus particles at a flat liquid-liquid interface. If the Janus boundary is completely sharp and smooth, no interface deformation occurs due to uniform wetting around the particles. However, if the neighboring particles possess different orientations or are pinned at specified angles, they interact due to the induced deformation at the fluid-fluid interface. The tendency to minimize high energy surface areas of the Janus particle distorts the contact line from a circular shape and results in attracting forces between the particles. We examine the energetic interactions among spherical Janus particles as a function of their separation distance, orientation angle, and their wettabilities. It is found that the extent of interface deformation strongly depends on the difference between the wettabilities of the two hemispherical sides. We show that the interface distortions at the near sides of the two spheres join, under appropriate conditions, to form an interfacial structure resembling a capillary bridge. Our calculation provides a detailed insight into the interface deformation and inter-particle forces that arise between randomly oriented Janus spheres before reaching their equilibrium orientation.

9:57AM G19.00010 The Short-range Capillary Force on Floating Objects , ANDONG HE, Nordic Institute for Theoretical Physics (NORDITA), KHOI NGUYEN, SHREYAS MANDRE, School of Engineering, Brown University — We develop a general method to study the capillary force between objects of arbitrary shape which float close to each other on an interface, a regime in which multipole expansion is not useful. The force is represented as a power series in the small distance between the objects, of which the leading-order is finite. For objects with size a much larger than the capillary length l_c, the force scales as $\sqrt{a/l_c}$ and the prefactor depends on the mean radius of curvature R at the closest points. After contact the objects roll and/or slide with respect to each other to locally maximize R. For smaller objects ($a \ll l_c$), the force scales as $(a/l_c)^{-1}\log(a/l_c)^{-2}$, and the prefactor depends only weakly on the shape and relative orientation of the objects.

Monday, November 19, 2012 8:00AM - 10:10AM –

Session G20 Turbulent Boundary Layers IV: Scaling, logarithmic layer 30A - Chair: Ron Panton, University of Texas

8:00AM G20.00001 Fluctuating Vorticity in Turbulent Boundary Layers , RON PANTON, University of Texas — Profiles of fluctuating vorticity from the channel flow DNS (Del Alamo, et al. (P of F **15**, L-41; JFM, **500**, p135, P of F, **18**) are correlated in Panton (Phys. Fluids, **21**, 2009). In the inner region, a two-term expansion represents the vorticity profiles; $\langle\omega\omega\rangle^{\#} = \langle\omega\omega\rangle^{\#}_{0} + \langle\omega\omega\rangle^{+}_{1} u_\tau/U_0$. The scaling $\langle\omega\omega\rangle^{\#}_{0} = \langle\omega\omega\rangle_0/(u_\tau^3 U_0/\nu^2)$ for inactive motions applies only to the streamwise and spanwise components. This term is zero for the normal vorticity component. The scrubbing of the inactive motions over the wall generates vorticity, which is a maximum at the wall, and diffuses to about $y^+ = 50$ before it decays. The fluctuating wall shear stress is due entirely to this motion, and the stress ratio (rms/mean) depends on *Re*. The second scaling $\langle\omega\omega\rangle^{+}_{1} = \langle\omega\omega\rangle_1/(u_\tau^4/\nu^2)$, the same scaling as the Reynolds shear stress, is active motions. These motions are zero at the wall, peak about $y^+ = 13$–20, and fall to zero about $y^+ = 400$. The outer region is correlated by a third scaling using the Kolmogorov time scale; $\langle\omega\omega\rangle/(u_\tau^3/\delta\ \nu)$. Matching between the inner and outer regions has an overlap law (common part) of $\sim C/y^+$ or $\sim C/Y$ for all components. In this paper DNS boundary layer data of Schlatter et al. (Phys. Fluids, **21,** 2009) is correlated in the manner previously used for channel flows.

8:13AM G20.00002 Von Karman re-visited[1] , DONALD M. MCELIGOT, U. Idaho, KEVIN P. NOLAN, Imperial College, EDMOND J. WALSH, U. Limerick — A number of authors have presented extended versions of the integral momentum equation, allowing for perturbations or fluctuations in the boundary layer. "Conventional wisdom" is that these added terms can be neglected and one can apply the von Karman version directly. For two-dimensional turbulent boundary layers at high Reynolds numbers, experience shows this assumption to be reasonable. However, recent examination of entropy generation in bypass transition with zero pressure gradient shows a term for turbulence energy convection can be important in determining the energy dissipation coefficient [Walsh et al., JFE 2011]. The present study employs the direct numerical simulations of Zaki and Durbin [JFM 2006] for bypass transition with streamwise pressure gradients to quantify the additional normal stress term when estimating the skin friction coefficient via a momentum balance. It is found that this term becomes noticeable in the pre-transitional laminar boundary layer and can exceed forty per cent of C_f in the transition region. Thus, it should be included in such calculations.

[1]Partly supported by U. S. DoE EPSCoR office

8:26AM G20.00003 The quest for the von Kármán constant , P.H. ALFREDSSON, R. ÖRLÜ, A. SEGALINI, Linne FLOW Centre, KTH Mechanics, Stockholm, Sweden — Already in 1930 von Kármán presented an expression for the mean velocity distribution in channel and pipe flows, that can be transformed in the today well known logarithmic velocity distribution. He was also able to obtain a value of 0.38 for the inverse of its slope, what we now know as the von Kármán constant (κ). In his case the value was obtained from pressure drop measurements and the, at the same time, formulated logarithmic skin friction law. Since then different values of κ have been suggested ranging from 0.37 to 0.44. Different approaches to determine κ have been suggested over the years, and also the range of the wall normal coordinate of the boundary layer over which the logarithmic law is valid have been debated. Not until independent measurements of the wall shear stress were available has there been a possibility to actually determine κ accurately from the measured mean velocity distribution. We discuss various pitfalls and error sources and based on a new straightforward method to determine κ, we use data from the literature to show that von Kármán's original suggestion of the value of κ seems to be valid also today.

8:39AM G20.00004 A universal logarithmic region in wall turbulence , IVAN MARUSIC, JASON MONTY, University of Melbourne, MARCUS HULTMARK, ALEXANDER SMITS, Princeton University — Considerable discussion over the past few years has been devoted to the question of whether the logarithmic region in wall turbulence is indeed universal. Here, we analyse recent experimental data in the Reynolds number range of nominally $2 \times 10^3 < Re_\tau < 6 \times 10^5$ for boundary layers, pipe flow and the atmospheric surface layer, and show that within the experimental uncertainty, the data support the existence of a universal logarithmic region. The results support the theory of Townsend (1976) and Perry & Chong (1982) that an inertial region requires both a logarithmic profile for the mean flow and the streamwise turbulence intensities. The experimental data are unique given the high Reynolds numbers presented and the fidelity of the measurement techniques where both the mean velocity and streamwise turbulence intensities are measured with the same instrument.

8:52AM G20.00005 Unified description of logarithmic profiles in a turbulent channel and pipe , ZHEN-SU SHE, XI CHEN, College of Engineering, Peking University, FAZLE HUSSAIN, Department of Mechanical Engineering, University of Houston — A similarity is discovered between the transports of the mean momentum and turbulent kinetic-energy, based on empirical analysis of the two balance equations in DNS data. It yields a new invariant distribution characterizing universal bulk flow dynamics in a channel or a pipe. The theory derives a logarithmic law for the mean kinetic-energy profile at high enough Reynolds numbers (Re). In particular, a Karman-like constant (0.8) for energy is obtained, which yields a quantitative explanation for a recent discovery of Hulkmark et al. (PRL, 2012) with right empirical constants. Together with the momentum Karman constant (0.45), we offer a unified description of the logarithmic distribution for both momentum and kinetic energy. Finally, the newly-found similarity governs also the temperature variations in Rayleigh-Benard convection, and the common log law originates from a sub-leading-order effect of turbulent transport in balancing the difference between turbulence production and dissipation.

9:05AM G20.00006 Spectral analogue of the law of the wall , GUSTAVO GIOIA, CARLO ZUNIGA ZAMALLOA, PINAKI CHAKRABORTY, Okinawa Institute of Science and Technology — We use a recently-proposed spectral model (Gioia et al., PRL, 2010) of the Reynolds shear stress in smooth wall-bounded, uniform turbulent flows to derive a scaling relation for the turbulent energy spectra. This scaling relation is the spectral analogue of Prandtl's scaling relation for the mean velocity profiles (the "law of the wall"). To test the scaling relation for the turbulent energy spectra, we use data from direct numerical simulations of channel flow.

9:18AM G20.00007 Mesolayer analysis in a turbulent boundary layer and DNS data , NOOR AFZAL, Retired Aligarh University, Aligarh 202002, India — The intermediate layer (mesolayer) in turbulent boundary layer has been analysed by the matched asymptotic expansions where matching is implemented by Izakson-Millikan-Kolmogorov hypothesis. The large-scale motions and very large scale motion are modifying the influences of the outer geometries, and most significantly near the locus of the peak in shear stress in the mesolayer. The mesolayer is formed by the interaction of inner and outer layer scales, whose length (time) scale is the geometric mean of the inner and outer length (time) scales, and is also proportional to Taylor micro length (time) scale. The mesolayer variable is proportional to inverse square root of appropriate friction Reynolds number, provided Reynolds number is large. It is shown that the shape factor and Reynolds shear maxima scale with mesolayer scale equivalent to Taylor micro length scale. Further, the turbulent bursting time period scales is shown to mesolayer time scale which is equivalent to Taylor micro time scale. The implications of mesolayer on higher order effects on skin friction law for lower Reynolds number have also been analyzed. The implications of shift origin are proposed by the Prandtl's transposition theorem, and consequently without any closure model.

9:31AM G20.00008 Model-based scaling and prediction of streamwise energy spectrum at high Reynolds numbers[1] , RASHAD MOARREF, Graduate Aerospace Laboratories, Caltech, USA, ATI S. SHARMA, Automatic Control & Systems Engineering, University of Sheffield, UK, JOEL A. TROPP, Computing & Mathematical Sciences, Caltech, USA, BEVERLEY J. MCKEON, Graduate Aerospace Laboratories, Caltech, USA — To better understand the behavior of wall-bounded turbulent flows at high Reynolds numbers, we study the Reynolds number scaling of the low-rank approximation to turbulent channel flows. Following McKeon and Sharma (J. Fluid Mech. 2010), the velocity is decomposed into propagating waves (with single streamwise and spanwise wavelengths and wave speed) whose wall-normal shapes are determined from the principal singular function of the corresponding resolvent operator. We identify three regions of wave parameters that induce intrinsic Reynolds number scales on the low-rank model, reveal the universal shape of the streamwise energy spectrum for the model subject to broadband forcing, and show that this model captures the dominant near-wall turbulent structures. The model-based streamwise spectrum is then shaped by optimal weight functions to match direct numerical simulations throughout the channel at low Reynolds numbers. Representation of the resulting weight functions using similarity laws facilities predictions of the streamwise energy spectra at high Reynolds numbers ($R_\tau \approx 10^3 - 10^6$) which are shown to agree closely with experiments.

[1]The support of AFOSR under grant FA 9550-09-1-0701 is gratefully acknowledged.

9:44AM G20.00009 Multi-layer prediction of mean velocity profiles in turbulent boundary layers , XI CHEN, College of Engineering, Peking University, FAZLE HUSSAIN, Department of Mechanical Engineering, University of Houston, ZHEN-SU SHE, College of Engineering, Peking University — A multi-layer prediction of the mean velocity profile (MVP) is developed for the zero pressure gradient (ZPG) turbulent boundary layer (TBL), in good agreement with empirical data over a wide range of the Reynolds number (Re). The theory builds on our model of the mixing length for channel and pipe flows, in which all of the physical parameters characterizing the viscous sublayer, buffer layer and bulk layer are held universal, as well as the Karman constant 0.45. The theory predicts a logarithmic law constant B of 6.5. The identified differences between the channel/pipe and TBL are the absence of a wall-confined central core layer and a fractional scaling of the total stress for the latter. Then, the theory yields an analytic expression for the wake function and friction coefficient in excellent agreement with measurements. In conclusion, a unified theory is presented for the MVPs of all canonical wall-bounded turbulent flows.

9:57AM G20.00010 A new scaling for the streamwise broadband turbulence intensity profiles of ZPG turbulent boundary layers , VIGNESHWARAN KULANDAIVELU, NICHOLAS HUTCHINS, The University of Melbourne, THE UNIVERSITY OF MELBOURNE TEAM — Turbulent boundary layers under zero pressure gradient are investigated experimentally with the aim of proposing a new scaling for the streamwise turbulence intensity. The streamwise intensity normalized by the inner and outer scales seems to collapse the profiles near the wall and in the wake region respectively. We here suggest a new scaling that aims to collapse these profiles across both the inner and outer regions. This is done by assuming a logarithmic variation between the viscous-scaling at the wall and outer scaling in the wake region. It is defined as $\hat{z} = \log_{10}\left(z^+/C\right)/\log_{10}\left(\delta^+/C\right)$, with $\hat{z} = 0$, at $z^+ = C$ and $\hat{z} = 1$, at $z^+ = \delta^+ = Re_\tau$. A very good collapse of the data is observed from $z^+ \approx 15$ to $z/\delta \approx 1$. The constant *"C"* is chosen to be 15 which signifies the inner normalised wall location z^+, where the peak in turbulence intensity is observed.

Monday, November 19, 2012 8:00AM - 10:10AM –

Session G21 Turbulence Simulation: Wall Bounded 30B - Chair: Oscar Flores, Universidad Carlos III de Madrid

8:00AM G21.00001 Slow Growth Formulation for DNS of Chemically Reacting Temporal Boundary Layers with Forcing , VICTOR TOPALIAN, TODD OLIVER, ROBERT MOSER, University of Texas at Austin — Extensions to a previously developed formulation for DNS of temporally evolving boundary layers are presented. The original formulation, which allows characterization of turbulence in a temporal boundary layer at a chosen stage of the development, uses a multiscale approach where the fast evolution of the turbulent fluctuations is simulated directly while the slow evolution of averaged quantities is modeled. Specifically, the source terms from slow evolution are modeled assuming self-similarity in the evolution of mean and RMS quantities. Here, the formulation is extended to enable DNS of chemically reacting boundary layers with forcing. These extensions are used to obtain DNS data for conditions similar to those observed in the boundary layer during athmospheric reentry of the NASA CEV. Data from this simulation will be used to inform turbulence model calibration and UQ. This work is supported by the Department of Energy [National Nuclear Security Administration] under Award Number [DE-FC52-08NA28615].

8:13AM G21.00002 Direct Numerical Simulation of two superposed viscous fluids in a channel with cavities on the wall , STEFANO LEONARDI, University of Puerto Rico at Mayaguez, PAOLO ORLANDI, Universita' degli Studi di Roma "La Sapienza" — Parallel shear flow of two viscous fluids has received much attention in the past. An instability associated with the jump in viscosity at the interface between two fluids has been observed and it depends on the wavenumber of the flow disturbance, on the depth ratio and the viscosity ratio of the fluids. In the present paper, we extend previous findings by performing Direct Numerical Simulations of two superposed viscous fluids over rectangular cavities in a channel. The interface between the two fluids is slightly above the crests plane of the cavities. This configuration represents a simplified and preliminary model of a Slippery Liquid-Infused Porous Surface (SLIPS) promising for drag reduction. Periodic boundary conditions apply in the streamwise and spanwise direction. A parametric study has been carried out varying the viscous ratio, the Reynolds number of the inner flow, the aspect ratio of the cavities and the depth of the near wall fluid. A finite difference code, based on a Runge Kutta and fractional step, has been used. Roughness on the walls has been modeled using the immersed boundary method. Frictional and form drag and transition to turbulence depend strongly on the cavity shape and on the viscous ratio.

8:26AM G21.00003 Aspect ratio effects in turbulent duct flows studied with DNS , R. VINUESA, IIT, Chicago, A. NOORANI, KTH, Stockholm, A. LOZANO-DURÁN, U. Politecnica de Madrid, P. SCHLATTER, KTH, Stockholm, P. FISCHER, Argonne National Lab, Chicago, H. NAGIB, IIT, Chicago — Three-dimensional effects present in turbulent duct flows, i.e., side-wall boundary layers and secondary motions, are studied by means of direct numerical simulations (DNS). The spectral element code Nek5000, developed by Fischer $et.\ al.$ (2008), is used to compute turbulent duct flows with aspect ratios 1 and 3 in streamwise-periodic boxes of length $25h$ (long enough to capture the longest streamwise structures). The total number of grid points is 28 and 62 million respectively, and the inflow conditions were adjusted iteratively in order to keep the same bulk Reynolds number at the centerplane $\left(Re_{b,c} = 2800\right)$ in both cases. Spanwise variations in wall shear, mean-flow profiles and turbulence statistics were analyzed with aspect ratio, and also compared with the 2D channel. The simulations were started from a laminar duct profile, and transition to turbulence was triggered by means of trip-forcing in the wall-normal direction, applied at the two horizontal walls. In addition, we developed a convergence criterion aimed at assessing the necessary averaging time T_A for converged statistics. We find that econdary motions present in duct flows require longer averaging times and the total shear-stress profile is not necessarily linear.

8:39AM G21.00004 Direct numerical simulation for turbulent channel flow at high Reynolds number[1] , MYOUNGKYU LEE, NICHOLAS MALAYA, ROBERT D. MOSER, University of Texas at Austin — Direct numerical simulation (DNS) is a powerful tool in the study of wall-bounded turbulent flows. Of particular focus is the scaling of flow statistics with respect to Reynolds number. Investigations of these scalings require data at higher Reynolds numbers, which is limited by available computational power. We have developed a new codebase, optimized for Petascale machines, in order to perform a DNS at higher Reynolds number ($Re_\tau \approx 5000$) than previously performed. We simulate a canonical channel flow, with two infinite parallel plates driven by a constant pressure gradient. The numerical scheme is a Fourier spectral representation in the streamwise and spanwise directions, with B-Splines in the inhomogeneous direction. We demonstrate agreement between this code and previous DNS results at lower Reynolds numbers. Finally, we present some preliminary statistics including the mean velocity profile and the intensity of the fluctuations.

[1]This work is supported by NSF PetaApps grants: OCI-0749223 and NSF PRAC Grant 0832634.

8:52AM G21.00005 Application of mean wall shear stress boundary condition to complex turbulent flows using a wall-modeled large eddy simulation[1] , MINJEONG CHO, JUNGIL LEE, HAECHEON CHOI, Seoul National University — The mean wall shear stress boundary condition was successfully applied to turbulent channel and boundary flows using large eddy simulation without resolving near-wall region (see Lee, Cho & Choi in this book of abstracts). In the present study, we apply this boundary condition to more complex flows where flow separation and redeveloping flow exist. As a test problem, we consider flow over a backward-facing step at $Re_h = 22860$ based on the step height. Turbulent boundary layer flow at the inlet ($Re_\theta = 1050$) is obtained using inflow generation technique by Lund et al. (1998) but with wall shear stress boundary condition. First, we prescribe the mean wall shear stress distribution obtained from DNS (Kim, 2011, Ph.D. Thesis, Stanford U.) as the boundary condition of present simulation. Here we give no-slip boundary condition at flow-reversal region. The present results are in good agreements with the flow statistics by DNS. Currently, a dynamic approach of obtaining mean wall shear stress based on the log-law is being applied to the flow having flow separation and its results will be shown in the presentation.

[1]Supported by the WCU and NRF programs

9:05AM G21.00006 Off-wall boundary conditions for turbulent flows obtained from buffer-layer minimal flow units[1] , RICARDO GARCIA-MAYORAL, BRIAN PIERCE, CTR, Stanford University, JAMES WALLACE, Burgers Program, University of Maryland — There is strong evidence that the transport processes in the buffer region of wall-bounded turbulence are common across various flow configurations, even in the embryonic turbulence in transition (Park et al., *Phys. Fl.* **24**). We use this premise to develop off-wall boundary conditions for turbulent simulations. Boundary conditions are constructed from DNS databases using periodic minimal flow units and reduced order modeling. The DNS data was taken from a channel at $Re_\tau = 400$ and a zero-pressure gradient transitional boundary layer (Sayadi et al., submitted to *J. Fluid Mech.). Both types of boundary conditions were first tested on a DNS of the core of the channel flow with the aim of extending their application to LES and to spatially evolving flows.*

[1]2012 CTR Summer Program

9:18AM G21.00007 Effect of thermal boundary condition on wall-bounded, stably-stratified turbulence , OSCAR FLORES, MANUEL GARCIA-VILLALBA, Universidad Carlos III de Madrid — The dynamics of stably stratified wall-bounded turbulent flows are of great importance for many engineering and geophysical problems. In some cases, like the stably stratified atmospheric boundary layer, it is unclear which is the most appropriate thermal boundary condition, i.e. constant temperature or constant flux at the ground. Here, we analyze the effect that this boundary condition has on the dynamics of turbulent motions in the near-wall region in the case of strong stable stratification. Two Direct Numerical Simulations of turbulent channels will be used, at $Re_\tau = u_\tau h/\nu = 560$ and $Ri_\tau = \Delta\rho g h/\rho_0 u_\tau^2 = 600 - 900$, which are described in detail in Flores & Riley (2011, Boundary-Layer Meteorol) and Garcia-Villalba & del Alamo (2011, Phys.Fluids). For this range of Reynolds and Richardson numbers, the near-wall region is intermittent, with patches of laminar flow embedded in the otherwise turbulent flow. It is in this regime where the differences between the constant temperature and the constant flux boundary conditions are expected to be larger, with the thermal boundary condition affecting how the local relaminarization of the flow takes place. This research has been supported by ARO, NSF and the German Research Foundation.

9:31AM G21.00008 The influence of viscosity stratification on boundary-layer turbulence[1] , JIN LEE, KAIST, SEO YOON JUNG, Imperial College London, HYUNG JIN SUNG, KAIST, TAMER A. ZAKI, Imperial College London — Direct numerical simulations of turbulent flows over isothermally-heated walls were performed to investigate the influence of viscosity stratification on boundary-layer turbulence and drag. The adopted model for temperature-dependent viscosity was typical of water. The free-stream temperature was set to 30°C, and two wall temperatures, 70°C and 99°C, were simulated. In the heated flows, the mean shear-rate is enhanced near the wall and reduced in the buffer region, which induces a reduction in turbulence production. On the other hand, the turbulence dissipation is enhanced near the wall, despite the the reduction in fluid viscosity. The higher dissipation is attributed to a decrease in the smallest length scales and near-wall fine-scale motions. The combined effect of the reduced production and enhanced dissipation leads to lower Reynolds shear stresses and, as a result, reduction of the skin-friction coefficient.

[1]Supported by the Engineering and Physical Sciences Research Council (Grant EP/F034997/1) and partially supported by the Erasmus Mundus Build on Euro-Asian Mobility (EM-BEAM) programme.

9:44AM G21.00009 Statistics of Stagnation Points in Turbulent Channel Flows with Wavy Walls , FABIAN HENNIG, MICHAEL GAUDING, JENS HENRIK GOEBBERT, NORBERT PETERS, RWTH Aachen University — We investigate the turbulent velocity field my means of instantaneous streamlines. The streamlines are partitioned into segments and decompose the velocity field in a non-arbitrary way. The segments are defined by extreme points based on the velocity magnitude. The boundaries of all streamline segments define a surface in space where the gradient of the projected velocity in streamline direction $\partial u/\partial s$ vanishes. This surface contains all local extreme points of the velocity magnitude. Such points also include stagnation points of the flow field, which are absolute minimum points of the turbulent velocity field. The properties of the $\partial u/\partial s = 0$ surface (and thus of stagnation points) are affected by local pressure gradients. Therefore, direct numerical simulations (DNS) of a turbulent flow with wavy walls, which induce complex pressure effects, are conducted. For the DNS a spectral element code is employed. The results have been validated against DNS and experimental data from literature. Based on the DNS the surface $\partial u/\partial s = 0$ is investigated in detail and its interaction with streamlines is visualized. The location and statistics of stagnation points with respect to the specific flow geometry are examined.

9:57AM G21.00010 Turbulent wall jets over rough surfaces , RAYHANEH BANYASSADY, UGO PIOMELLI, Queen's University, Kingston (ON), Canada — The effects of surface roughness on plane and radial turbulent wall jets have been studied using large-eddy simulation. The Reynolds number is $18,800$ (based on the bulk velocity and diameter of the impinging jet) for the radial case, and $7,500$ (based on the jet velocity and height) for the plane one. To represent the random roughness elements a virtual sandpaper model and an immersed-boundary method (IBM) based on the volume-of-fluid (VOF) approach are used. The roughness Reynolds numbers in both simulations are in the transitionally rough regime, $5 < k^+ < 70$. The grid refinement study showed that the plane and radial wall jet simulations require 15 and 22 million grid points, respectively. The results are validated with available literature. The mean flow shows that surface roughness decreases maximum velocity of the wall jet and increases the wall jet characteristics length (distance from the wall where the velocity decreases to half of the maximum velocity). The rate of decay of maximum velocity and growth rate of the wall jet are not significantly affected by the surface roughness. An analysis of the effect of surface roughness on the magnitude and shape of the Reynolds stresses profiles is being carried out.

Monday, November 19, 2012 8:00AM - 10:10AM –

Session G22 Turbulence Mixing I: Mixing in Jets 30C - Chair: Godfrey Mungal, Stanford University

8:00AM G22.00001 Density and Velocity Ratios Effects on the Structure of Transverse Jets in Supersonic Crossflow[1] , MIRKO GAMBA, University of Michigan, VICTOR A. MILLER, M. GODFREY MUNGAL, Stanford University — It is generally accepted that the jet-to-crossflow momentum flux ratio, J, is the primary parameter describing the structure, penetration and mixing properties of transverse jets. The interplay between density and velocity ratios, that combined define J, on these properties is, however, seldom considered nor fully understood. The current experimental work explores this interplay on transverse underexpanded sonic jets in a supersonic nitrogen crossflow ($M = 2.25$, $T = 480\ K$). A single-excitation, dual-band detection PLIF imaging scheme of toluene seeded into the crossflow is used to mark the crossflow fluid mixing into the transverse jet fluid and to determine the local fluid temperature. Different values of density and velocity ratios, while maintaining a constant value of J (equal to 2.4), are investigated by injecting gases with different molecular weights. Notwithstanding the fact that all cases have the same value of J and some similarity on the penetration characteristics exists, the emerging picture of the instantaneous turbulent structure of the flow indicates that the dynamics of entrainment and local mixing might be altered by low (high) values of the density (velocity) ratio compared to the corresponding case at high (low) values.

[1]Sponsored by DoE PSAAP at Stanford University

8:13AM G22.00002 Numerical Study on Cryogenic Coflowing Jets under Transcritical Conditions , HIROUMI TANI, Japan Aerospace Exploration Agency, SUSUMU TERAMOTO, KOJI OKAMOTO, University of Tokyo, NOBUHIRO YAMANISHI, Japan Aerospace Exploration Agency — A numerical and experimental study is presented on cryogenic coflowing jets under transcritical conditions for a better understanding of the propellant mixing in supercritical-pressure rocket engines. The major concerns are dominant flow structures in the mixing of cryogenic coflowing jets under transcritical conditions. Experimentally, in advance of detailed numerical simulations, cryogenic nitrogen/gaseous nitrogen coaxial jets were visualized by the backlighting photography technique. It was observed that a dense nitrogen core has a shear-layer instability near the injector exit and eventually breaks up into large lumps which dissolve and fade away downstream. In numerical simulations, LES technique was employed for more detailed discussion on the flow structures. LES of a cryogenic nitrogen/gaseous nitrogen coflowing plane jet was conducted with the same density and velocity ratios of inner/outer jets as the experiments. As observed in the experiments, the shear-layer instability in the inner mixing layers is predominant near the injector exit. After roll-up and paring, the shear-layer instability waves become large-scale vortices. They cause coherent vortex structures which become dominant in the downstream and break the dense core into lumps. Strouhal numbers of the shear-layer instability and the dense lump shedding in the numerical simulations were comparable to those measured in the experiments, respectively.

8:26AM G22.00003 Large-eddy-simulation and measurements of turbulent transport and mixing in a confined rectangular jet , JAMES HILL, KATRINE NILSEN, BO KONG, MICHAEL OLSEN, RODNEY FOX, Iowa State University — Large-eddy simulations of transport and mixing of a passive scalar were performed for a confined rectangular liquid jet (Re = 20,000) and compared to results of simultaneous particle image velocimetry and planar laser induced fluorescence measurements. Reasonably good agreement was obtained for single-point statistics as well as for two-point correlation functions of the turbulent velocity, scalar, and joint velocity-scalar fields. Of particular interest was the determination of the diagonal and off-diagonal components of the turbulent diffusivity tensor, the mis-alignment of turbulent fluxes and mean gradients, and determination of the turbulent Schmidt number, with reasonably good agreement between the simulations and experiments. Some of the results are consistent with the measurements of Tavoularis & Corrsin (JFM 104, 331-367 (1981)) for a homogeneous turbulent shear flow with a uniform mean temperature gradient. In our case we find the ratio of diffusivities to range from 1 to 2 (the latter value in agreement with Tavoularis), Sc_t generally between 0.5 and 1, and the angle between turbulent flux and mean scalar gradient from 120 to 150 degrees.

8:39AM G22.00004 Gradient trajectory analysis of the scalar superlayer in a jet flow[1] , MARKUS GAMPERT, PHILIP SCHAEFER, NORBERT PETERS, RWTH Aachen — Based on planar high-speed Rayleigh scattering measurements of the mass fraction of propane discharging from a turbulent round jet into co-flowing carbon dioxide at nozzle based Reynolds numbers Re_0=3,000-8,600, we investigate the *scalar superlayer*. The latter is located between the fully turbulent part of the jet and the outer flow and has the so called turbulent/non-turbulent interface embedded within it. It is termed in analogy to the laminar superlayer introduced by Corrsin and Kistler (NACA Report 1244, 1955). Using scalar gradient trajectories, we partition the turbulent scalar field into the afore mentioned three regions according to an approach developed by Mellado et al. (J. Fluid Mech. 626:333-365, 2009) based on which we in a next step investigate conditioned zonal statistics of the scalar pdf as well as the scalar difference along the trajectory and its mean scalar value. Finally, we relate our results for the scalar superlayer on the one hand to the findings made in other experimental and numerical studies of the turbulent/non-turbulent interface and discuss them on the other hand in the context of the flamelet approach in turbulent non-premixed combustion.

[1]This work was funded by the Cluster of Excellence "Tailor-Made Fuels from Biomass," which is funded by the Excellence Initiative of the German federal state governments to promote science and research at German universities.

8:52AM G22.00005 Variable Density Turbulent Jet Mixing , SERGIY GERASHCHENKO, KATHY PRESTRIDGE, Los Alamos National Laboratory — Variable density mixing arises due to differences in molecular weights of the mixing fluids, or due to compressibility effects. A detailed understanding of the mixing processes has important consequences for many scientific and engineering systems such as inertial confinement fusion, atmospheric flows and oceans, or supernovae explosions. A new experiment has been developed at LANL to study the fundamental statistical properties of variable density turbulence that decays in time in subsonic incompressible flows. Initial experimental results are presented of a heavy fluid (sulfur hexafluoride gas) turbulent jet injected into coflowing air of lower turbulence. Buoyancy-mediated mixing is investigated at two Atwood numbers: 0 and 0.6, for a range of Reynolds numbers. The velocity and density fields are measured with simultaneous Particle Image Velocimetry and Planar Laser Induced Fluorescence. The fundamental statistical characteristics of the mixing important for modeling these flows such as spreading rate, mass flux, density self-correlation, kinetic energy flux, and turbulence decay rate are examined.

9:05AM G22.00006 ABSTRACT WITHDRAWN –

9:18AM G22.00007 Large eddy simulation study of mixing in stratified jets , NIRANJAN GHAISAS, DINESH SHETTY, STEVEN FRANKEL, School of Mechanical Engineering, Purdue University — The structure and dynamics of a horizontally injected round turbulent buoyant jet are studied using Large Eddy Simulation (LES). A high-order accurate numerical procedure and different constant coefficient and dynamic eddy-viscosity sub-grid scale (SGS) models are used. The numerical procedure and SGS models are validated by conducting simulations at previous experimental flow conditions (characterized by Reynolds number, Re, and Richardson number, Ri), and comparing with the existing experimental results. The experimental results are then supplemented by simulations at different Re and Ri. Previous studies have shown that interaction between buoyancy and turbulence in this configuration leads to suppressed mixing in stably stratified regions and enhanced turbulence and mixing in unstably stratified regions. The ability of different SGS models to capture this phenomenon is studied by examining the jet trajectory, decay of the center-line velocity, radial spread and turbulent kinetic energy budgets at different axial locations. Schmidt number dependence of the results is also analyzed. Finally, the existence of secondary flows which lead to a plume-like vertical motion in addition to the primary horizontal injection is discussed and quantified.

9:31AM G22.00008 Rapid Confined Mixing with Transverse Jets Part 1: Single Jet , DAVID SALAZAR, DAVID FORLITI, Sierra Lobo Inc., Contractor for the Air Force Research Laboratory — Transverse jets have been studied extensively due to their relevance and efficiency in fluid mixing applications. Gas turbine burners, film cooling, and chemical reactors are some examples of rapid transverse jet mixing. Motivated by a lack of universal scaling laws for confined and unconfined transverse jets, a newly developed momentum transfer parameter was found to improve correlation of literature data. Jet column drag and entrainment arguments for momentum transfer are made to derive the parameter. A liquid-phase mixing study was conducted to investigate confined mixing for a low number of jets. Planar laser induced fluorescence was implemented to measure mixture fraction for a single confined transverse jet. Time-averaged cross-sectional images were taken with a light sheet located three diameters downstream of transverse injection. A mixture of water and sodium fluorescein was used to distinguish jet fluid from main flow fluid for the test section images. Image data suggest regimes for under- and overpenetration of jet fluid into the main flow. The scaling parameter is found to correlate optimum unmixedness for multiple diameter ratios at a parameter value of 0.75. Distribution A: Public Release, Public Affairs Clearance Number: 12655.

9:44AM G22.00009 Rapid Confined Mixing Using Transverse Jets Part 2: Multiple Jets , DAVID FORLITI, DAVID SALAZAR, Sierra Lobo Inc., Contractor for the Air Force Research Laboratory — An experimental study has been conducted at the Air Force Research Laboratory at Edwards Air Force Base to investigate the properties of confined mixing devices that employ transverse jets. The experiment considers the mixing of water with a mixture of water and fluorescein, and planar laser induced fluorescence was used to measure instantaneous mixture fraction distributions in the cross section view. Part one of this study presents the scaling law development and results for a single confined transverse jet. Part two will describe the results of configurations including multiple transverse jets. The different regimes of mixing behavior, ranging from under to overpenetration of the transverse jets, are characterized in terms of a new scaling law parameter presented in part one. The level of unmixedness, a primary metric for mixing device performance, is quantified for different jet diameters, number of jets, and relative flow rates. It is apparent that the addition of a second transverse jet provides enhanced scalar uniformity in the main pipe flow cross section compared to a single jet. Three and six jet configurations also provide highly uniform scalar distributions. Turbulent scalar fluctuation intensities, spectral features, and spatial eigenfunctions using the proper orthogonal decomposition will be presented. Distribution A: Public Release, Public Affairs Clearance Number: 12656.

9:57AM G22.00010 Two-point statistics for turbulent relative dispersion in quasi-two-dimensional turbulent jets , JULIEN R. LANDEL, University of Cambridge, DAMTP, C.P. CAULFIELD, DAMTP & BPI, University of Cambridge, ANDREW W. WOODS, BPI, University of Cambridge — The study of turbulent jets in relatively enclosed geometries is relevant to 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. To measure turbulent relative dispersion in the streamwise and cross-stream directions of quasi-two-dimensional jets, we propose a Lagrangian-particle-tracking technique which we name virtual particle tracking (VPT). We seed virtual massless passive tracer particles in the velocity field of a flow measured experimentally using particle image velocimetry. These virtual particles evolve as point passive tracers in the time-dependent velocity field and can be tracked in time and space. After presenting the VPT technique, we show the time-evolution of two-point statistics, such as the distance between two virtual particles, measured from the evolution of large virtual particle clusters seeded in the flow of quasi-two-dimensional jets. We also compare the results given by VPT with a variety of other techniques. We find that the dispersion properties differ significantly between the large scale eddies and the high-speed sinuous core observed in the flow of quasi-two-dimensional jets. As a result, we observe large streamwise dispersion and a significant amount of tracers can be transported faster than the speed predicted by a simple top-hat advection model in the jet.

Monday, November 19, 2012 8:00AM - 9:57AM –

Session G23 General Fluids I 30D - Chair: Patrick Weidman, University of Colorado

8:00AM G23.00001 Obliquely-intersecting Hiemenz stagnation-point flows , PATRICK WEIDMAN, University of Colorado — The interaction of two obliquely-intersecting Hiemenz stagnation point flows normal to a flat plate is studied. One flow of strain rate a aligned along the x-axis intersects the other of strain rate b at angle Phi from the x-axis. When transformed to principal axes, a similarity reduction of the Navier-Stokes equation yields two ordinary differential equations with coefficients depending on the strain rate ratio Sigma = b/a and Phi. These equations are then tranformed to Howarth's equations via a two-parameter mapping. The large-Sigma asymptotic behavior of solutions for the limiting angles Phi = 0 and 90 deg. are determined. Numerical solutions of the principal axes equations for values of Sigma in the saddle-point and nodal-point regions at Phi = 0, 15, 30, 45, 60, 75 and 90 deg. compare precisely with those obtained from the two-parameter mapping. Plots of the wall shear stress parameters, the normalized displacement thicknesses and sample velocity profiles are presented.

8:13AM G23.00002 A numerical study of the motion of a neutrally buoyant cylinder in two dimensional shear flow[1] , TSORNG-WHAY PAN, Department of Mathematics, University of Houston, Houston, TX 77204, USA, SHIH-LIN HUANG, SHIH-DI CHEN, 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 cylinder of circular or elliptic shape in two dimensional shear flow of a Newtonian fluid by direct numerical simulation. The numerical results are validated by comparisons with existing theoretical, experimental and numerical results, including a power law of the normalized angular speed versus the particle Reynolds number. The centerline between two walls is an expected equilibrium position of the cylinder mass center in shear flow. When placing the particle away from the centerline initially, it migrates toward another equilibrium position for higher Reynolds numbers due to the interplay between the slip velocity, the Magnus force, and the wall repulsion force.

[1]T-W Pan acknowledges the support by the US NSF and S-L Huang, S-D Chen, C-C Chu, C-C Chang acknowledge the support by the National Science Council of Taiwan, ROC.

8:26AM G23.00003 An analysis of the "accidental painting" technique of D.A. Siqueiros: the Rayleigh Taylor instability as a tool to create explosive textures , S. ZETINA, R. ZENIT, Universidad Nacional Autonoma de Mexico — In the spring of 1936, the famous Mexican muralist David Alfaro Siqueiros organized an experimental painting workshop in New York: a group of artists focused in developing painting techniques through empirical experimentation of modern and industrial materials and tools. Among the young artists attending the workshop was Jackson Pollock. They tested different lacquers and a number of experimental techniques. One of the techniques, named by Siqueiros as a "controlled accident," consisted in pouring layers of paint of different colors on top of each other. After a brief time, the paint from the lower layer emerged from bottom to top creating a relatively regular pattern of blobs. This technique led to the creation of explosion-inspired textures and catastrophic images. We conducted an analysis of this process. We experimentally reproduced the patterns "discovered" by Siquieros and analyzed the behavior of the flow. We found that the flow is driven by the well-known Rayleigh Taylor instability: different colors paints have different densities; a heavy layer on top of a light one is an unstable configuration. The blobs and plumes that result from the instability create the aesthetically pleasing patterns. We discuss the importance of fluid mechanics in artistic creation.

8:39AM G23.00004 Data Reduction Modeling of a Graphite Nitridation Experiment[1] , PAUL BAUMAN, ROBERT MOSER, The University of Texas at Austin — In this work, we present a computational study of a flow tube experiment in order to infer reaction parameters for graphite nitridation. This study builds off previous work where it was determined existing, simplified models were inadequate to meaningfully inform parameters of the nitridation reaction of interest. We construct a two-dimensional representation of the experimental setup and model the flow using a reacting low-Mach number approximation to the Navier-Stokes equations. We use a stabilized finite element method to approximate the mathematical model. To solve the inverse problem for the reacting parameters, we employ a Bayesian approach in order to produce probability distributions that can used to quantify uncertainty in models where surface nitridation is important. Such models include the surface ablation of thermal protection systems of reentry vehicles.

[1]This material is based in part upon work supported by the Department of Energy [National Nuclear Security Administration] under Award Number [DE-FC52-08NA28615].

8:52AM G23.00005 Fluid flow and ponding on elastic beams , JEROME NEUFELD, BP Institute, Department of Earth Sciences, Department of Applied Mathematics and Theoretical Physics, University of Cambridge, RACHAEL BONNEBAIGT, Department of Applied Mathematics and Theoretical Physics, University of Cambridge — Fluids propagate over floating, elastically deformable surfaces in a variety of industrial, environmental, and geological settings, and can exhibit a wide variety of transient and steady-state behaviours. Here we show that, for a sufficiently localised release, steady states exist where a dense, viscous fluid propagates along a beam floating on a relatively light, inviscid ocean. The steady-state radius, fluid depth, and elastic deformation can be characterised solely as functions of the beam's bending length scale, and the relative densities of ocean and fluid. In contrast, for releasees of relatively light fluid no steady states exist, and a transient solution is instead found.

9:05AM G23.00006 Void collapse under distributed dynamic loading near material interfaces[1] , GALINA SHPUNTOVA, JOANNA AUSTIN, University of Illinois at Urbana-Champaign — Collapsing voids cause significant damage in diverse applications from biomedicine to underwater propulsion to explosives. While shock-induced void collapse has been studied extensively, less attention has been devoted to stress wave loading, which will occur instead if there are mechanisms for wave attenuation or if the impact velocity is relatively low. A set of dynamic experiments was carried out in a model experimental setup to investigate the effect of acoustic heterogeneities in the surrounding medium on void collapse. Two tissue-surrogate polymer materials of varying acoustic properties were used to create flowfield geometries involving a boundary and a void. A stress wave, generated by projectile impact, triggered void collapse in the gelatinous polymer medium. When the length scales of features in the flow field were on the same order of magnitude as the stress wave length scale, the presence of the boundary was found to affect the void collapse process relative to collapse in the absence of a boundary. This effect was quantified for a range of geometries and impact conditions using a two-color, single-frame particle image velocimetry technique.

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

9:18AM G23.00007 Fog-harvesting Mesh Surfaces , KYOO-CHUL PARK, Department of Mechanical Engineering, Massachusetts Institute of Technology, SHREERANG S. CHHATRE, SIDDARTH SRINIVASAN, ROBERT E. COHEN, Department of Chemical Engineering, Massachusetts Institute of Technology, GARETH H. MCKINLEY, Department of Mechanical Engineering, Massachusetts Institute of Technology — Fog represents a large, untapped source of potable water, especially in arid climates. Various plants and animals use morphological as well as chemical features on their surfaces to harvest this precious resource. In this work, we investigate the influence of surface wettability, structural length scale, and relative openness of the weave on the fog harvesting ability of mesh surfaces. We choose simple woven meshes as a canonical family of model permeable surfaces due to the ability to systematically vary periodicity, porosity, mechanical robustness and ease of fabrication. We measure the fog collecting capacity of a set of meshes with a directed aqueous aerosol stream to simulate a natural foggy environment. Further, we strive to develop and test appropriate scalings and correlations that quantify the collection of water on the mesh surfaces. These design rules can be deployed as an a priori design chart for designing optimal performance meshes for given environmental/operating conditions.

9:31AM G23.00008 Inertial Lubrication Theory , MEDERIC ARGENTINA, NICOLAS ROJAS, University Nice Sophia Anitpolis, ENRIQUE CERDA, Universidad Santiago de Chile, ENRIQUE TIRAPEGUI, Universidad de Chile — Thin fluid films can have surprising behavior depending on the boundary conditions enforced, the energy input and the specific Reynolds number of the fluid motion. Here we study the equations of motion for a thin fluid film with a free boundary and its other interface in contact with a solid wall. Although shear dissipation increases for thinner layers and the motion can generally be described in the limit as viscous, inertial modes can always be excited for a sufficiently high input of energy. We derive the minimal set of equations containing inertial effects in this strongly dissipative regime.

9:44AM G23.00009 An Experimental Study of Soft Lubrication , QIANHONG WU, THOMAS GACKA, Villanova University, RUNGUN NATHAN, Penn State Berks, LI-ZHU WU, The Chinese Academy of Sciences, CBMSS TEAM[1] — Lift generation in soft porous media, as a planing surface glides over it, is a new topic in porous media flow with superior potential for lubrication and squeeze damping. This paper presents the first experimental study of this phenomenon. The experimental setup consists of a running conveyer belt covered with a soft porous sheet, and a stationary instrumented inclined planar upper board. Twelve pressure transducers mounted on the upper board captured the pore pressure generation, while a load cell was used to capture the total lifting force, arising from both the pore pressure and the compression of the solid fibers. One finds that the pore pressure distribution is consistent with theoretical predictions (Feng and Weinbaum, JFM, 2000; Wu et al., MSSE, 2006, 2011), and depends on the running belt velocity, U, the mechanical properties of the porous material, and the compression ratios of the porous layer. For a typical trial (h_2/h_1=5,h_2/h_0=1, U=3.8 m/s, where h_2, h_1, and h_0 are the leading edge, trailing edge, and undeformed porous layer thicknesses, respectively), 68% of the lifting force was generated by the pore pressure. It conclusively demonstrates the validity of using soft porous materials for super lubrication. applications.

[1]Villanova Cellular Biomechanics and Sports Science Laboratory

Monday, November 19, 2012 8:00AM - 10:10AM –

Session G24 Aerodynamics II 30E - Chair: Daniel Maynes, Brigham Young University

8:00AM G24.00001 How Cristiano Ronaldo performs his knuckleball? , CAROLINE COHEN, BAPTISTE DARBOIS TEXIER, LadHyX, DAVID QUERE, PMMH, ESPCI, CHRISTOPHE CLANET, LadHyX — A soccer ball kicked at very low spin can exhibit a zigzag trajectory. Along its straight path, the ball deviates laterally from about 0.2 m, that is to say one ball diameter. One zig zag happens as the ball travelled about 15 m. As the deviation direction seems unpredictable, this effect is highly annoying for goalkeepers. That why Cristiano Ronaldo and many soccer players are looking for this phenomenon. Those trajectories called knuckleballs are also observed on volleyball and baseball. We study experimentally indoor knuckleballs for different balls varying from soccer balls to smooth spheres. We show that knuckle effect doesn't derive from ball deformations at foot impact or ball seams. Actually, aerodynamic lift forces on a smooth sphere are fluctuating and are responsible for knuckleballs. From this study, we deduce side force intensity exerted on smooth spheres and sport balls for typical game Reynolds number ($Re \sim 10^4 - 10^6$). Finally we discuss required conditions to observe a knuckleball on the sport field.

8:13AM G24.00002 Nuances between flags fluttering in horizontal and vertical flows , EMMANUEL VIROT, PASCAL HEMON, XAVIER AMANDOLESE, LadHyX, Ecole Polytechnique — When placed in a constant-velocity flow, a flag starts suddenly to flutter above a critical flow velocity. It appears that the same flag hanged horizontally or vertically exhibits different behaviors. The critical velocity is higher in the vertical case, suggesting an important role of gravity. To investigate the influence of length, width and thickness of paper flags, we perform experiments in both horizontal and vertical wind tunnels. The involvement of elasticity, fluid friction-induced tension or gravity-induced tension on the flag flutter will be discussed. The flag can be also a mean to harvest the wind kinetic energy: in a preliminary approach, we measure and discuss periodic forces that a horizontal flapping flag produces on his shaft.

8:26AM G24.00003 Why historical east deviation experiments are so difficult to perform? , BAPTISTE DARBOIS TEXIER, LadHyX - PMMH, ESPCI, CAROLINE COHEN, LadHyX, DAVID QUERE, PMMH, ESPCI, CHRISTOPHE CLANET, LadHyX — From the 17th to the 19th century, a big deal was to prove Earth rotation existence. For this purpose, numerous experimental physicists from Borelli in 1668 to Reich in 1832 tried to measure an eastward deviation of a falling sphere. Reich performed 106 falling experiments on a 158.5 m deep mine pit. The mean eastward deviation of its experiments is 2.8 cm. This value corresponds exactly with the theoretical one predicted by Laplace and Gauss expression at Freiberg latitude where experiments were conducted. While Reich took extreme precautions to perform its experiments, the dispersion on its results is very important. Actually aerodynamic lift forces on a smooth sphere made its free fall non perfectly straight. We understand Reich's results dispersion considering fluctuating lift forces intensity on a smooth sphere at those Reynolds numbers ($Re \sim 10^4 - 10^5$). This study provides a criterion above which we can distinguish between lift force and Coriolis deviation during a free fall experiment.

8:39AM G24.00004 Linearized boundary conditions at a rough surface[1] , PAOLO LUCHINI, Università di Salerno - DIIN — Linearized boundary conditions are a common numerical tool in any flow problems where the solid wall is nominally flat but the effects of small roughness of height ϵ are being investigated. Typical are receptivity problems in aerodynamic transition prediction or turbulent flow control. However, two distinguished mathematical limits have to be considered: a "shallow" limit, where the linearized boundary condition properly applies, involving a family of surfaces that become smoother and smoother as $\epsilon \to 0$, and a "small" limit, more closely representative of usually encountered roughness, whose family of surfaces remain geometrically similar to themselves (in particular, retain their slope) as $\epsilon \to 0$. A connection between the two limits will be established through an analysis of their asymptotic behaviour. As a result, the correct effect of the surface in the "small" limit, obtained through a numerical solution of the Stokes equation, will be recast as an equivalent linearized boundary condition modified by a suitable *protrusion coefficient* (related to the *protrusion height* used years ago in the study of riblets). Quantitative numerical examples of such protrusion coefficients will be provided.

[1]part of the RECEPT EC-FP7 project

8:52AM G24.00005 On the lift induced drag in viscous flows , RENATO TOGNACCINI, Università di Napoli Federico II, CLAUDIO MARONGIU, CIRA (Italian Aerospace Research Center), MAKOTO UENO, JAXA (Japan Aerospace Exploration Agency) — As stated by Spalart (*JFM*, 2008): "An ambition which will have to wait is a rigorous definition of induced drag in viscous flows." The idea that there is a link between the aerodynamic force and the Lamb vector, defined as the cross product of fluid vorticity and velocity dates back to Prandtl. Saffman ("Vortex Dynamics," 1992) and, more recently, Wu J.-Z. *et al.* (*JFM*, 2007) suggested an expression of the lift induced drag in terms of vortex force (the volume integral of the Lamb vector). In this paper we analyze the *steady* incompressible flow around a 3D lifting body at high Reynolds numbers. The suggested connection between vortex force and induced drag is discussed in detail. In particular, a rigorous definition of the lift induced drag in viscous flows without ambiguities is proposed. A numerical experiment: the analysis of the flow around an elliptic wing will confirm the theoretical analysis. The aerodynamic force and its lift and drag components are computed by integration of the Lamb vector field as obtained by a numerical solution and will be compared with classical expressions.

9:05AM G24.00006 Drag and near wake characteristics of flat plates normal to the flow with fractal edge geometries , JOVAN NEDIC, Imperial College London, BHARATH GANAPATHISUBRAMANI, University of Southampton, CHRISTOS VASSILICOS, Imperial College London — Past results have suggested that the coefficient of drag and shedding frequencies of regular polygon plates all fall within a very narrow band of values. In this study, we introduce a variety of length-scales into the perimeter of a square plate and study the effects this has on the wake characteristics and overall drag. The perimeter of the plate can be made as long as allowed by practical constraints with as many length-scales as desired under these constraints without changing the area of the plate. A total of eight fractal-perimeter plates were developed, split into two families of different fractal dimension all of which had the same frontal area. It is found that by increasing the number of fractal iterations, thus the perimeter, the drag coefficient increases by up to 10%. For the family of fractal plates with the higher dimension, it is also found that when the perimeter increases above a certain threshold the drag coefficient drops back again. Furthermore, the shedding frequency remains the same but the intensity of the shedding decreases with increasing fractal dimension. The size of the wake also decreases with increasing fractal dimension and has some dependence on iteration without changing the area of the plate.

9:18AM G24.00007 Slipping through the water: A study of superhydrophobic hydrofoils[1] , ROBERT DANIELLO, University of Massachusetts, Amherst, KIERSTIN DEL VALLE[2], None, JONATHAN ROTHSTEIN, University of Massachusetts, Amherst — Superhydrophobic surfaces which are chemically hydrophobic with micron or nanometer scale surface features have been studied for their ability to produce a slip interface which has been shown to affect drag, separation, lift, and vortex dynamics. In this talk, we will consider an experimental study of the effect of slip on lift, drag and stall of hydrofoils with a slip-producing superhydrophobic coating. Direct force measurements of lift and drag will be presented for a series of superhydrophobic and no-slip hydrofoils over a range of Reynolds numbers $3500 < Re < 35000$ and angles of attack from 0 to stall. Effects of slip on the boundary layer, separation and stall will be considered with particle image velocimetry.

[1]The authors would like to thank NSF for funding this research under grant CBET-0967531
[2]REU student, Webb Institute

9:31AM G24.00008 Flow over slippery liquid-infused porous surfaces , BRIAN ROSENBERG, GILAD ARWATZ, JESSICA SHANG, ALEXANDER SMITS, Princeton University — Slippery liquid-infused porous surfaces (SLIPS) demonstrate remarkable liquid repellency in addition to high pressure stability and rapid self-healing [T.S. Wong et al., Nature 2011]. The SLIPS surface consists of a thin lubricating film, locked in place on a nano-textured membrane, that permits mobility of the surrounding liquid at the interface. The relaxation of the no-slip boundary condition, in addition to their robustness in high-stress environments, means that these surfaces have promise to reduce drag in engineering flows. Here, we investigate the response of SLIPS in the laminar, transitional, and turbulent flow regimes. Experiments are performed in a Taylor-Couette apparatus, with water as the working fluid, over a wide Reynolds number range and with varying lubricant viscosities. We assess the skin friction properties of SLIPS surfaces and compare it to those of untreated surfaces.

9:44AM G24.00009 Drag Control through Wrinkling on Curved Surfaces[1] , DENIS TERWAGNE, Department of Civil and Environmental Engineering, Massachusetts Institute of Technology, MA, USA, PEDRO REIS, Department of Mechanical Engineering and Department of Civil and Environmental Engineering, Massachusetts Institute of Technology, MA, USA — We present the results of an experimental investigation on the wrinkling of positively curved surfaces and explore their use towards drag reduction applications. In our precision desktop-scale experiments we make use of rapid prototyping techniques to cast samples with custom geometry and material properties out of silicone-based rubbers. Our structures consist of a thin stiff shell that is chemically bonded to a thicker soft substrate. The substrate contains a spherical cavity that can be depressurized, under controlled volume conditions, to compress the ensemble structure. Under this compressive loading, the initially smooth outer-shell develops complex wrinkling patterns. We systematically characterize and quantify the morphology of the various patterns and study the phase diagram of the system. We consider both geometric and material quantities in the parameter space. Moreover, since the wrinkling patterns can be actuated dynamically using a pressure signal, we systematically characterize the aerodynamic behavior of our structures in the context of fluid drag reduction. An added advantage of the novel mechanism we introduce is that it allows for both dynamic switching and tuning of the surface morphology, thereby opening paths for drag control.

[1]D.T. thanks the B.A.E.F., the Fulbright Program and the WBI.World grants program for financial support.

9:57AM G24.00010 Vorticity Transport on a Rotating Blade[1] , CRAIG WOJCIK, JAMES BUCHHOLZ, University of Iowa — The development of the leading-edge vortex (LEV) is investigated on the suction surface of a rectangular flat plate undergoing a starting rotation in a quiescent fluid for angles of attack between 25 and 45 degrees. For blade aspect ratios of 2 and 4, the LEV is shown to be compact and quasi-stationary at inboard regions of the blade, consistent with the results of some other recent investigations. A salient feature of this flow is a region of opposite-sign vorticity generated on the blade beneath the LEV which is observed to become partially entrained into the LEV. A detailed vorticity transport analysis on the LEV has revealed that the resulting annihilation of vorticity is an important mechanism regulating LEV circulation, and therefore its stability. A parametric study is discussed, which elucidates the roles of shear layer vorticity flux, spanwise flow, vortex tilting, and annihilation on the evolution of LEV circulation with changes in azimuthal position, blade aspect ratio, spanwise position, and Reynolds number.

[1]This work was supported by the National Science Foundation (EPSCoR grant EPS1101284), the Air Force Office of Scientific Research (grant FA9550-11-1-0019), and IIHR - Hydroscience & Engineering.

Monday, November 19, 2012 8:00AM - 10:10AM –

Session G25 Flow Control: Wakes 31A - Chair: Philippe Lavoie, University of Toronto

8:00AM G25.00001 Optimal open-loop control of the recirculation length in the wake of a cylinder , EDOUARD BOUJO, FRANCOIS GALLAIRE, LFMI, EPFL, Lausanne, Switzerland — We consider steady equilibrium solution of the 2-D Navier-Stokes equations describing the flow around a circular cylinder. It is well known that, for moderate Reynolds numbers, the recirculation length is an increasing function of the Reynolds number. We first express the recirculation length as a cost-functional and then use a Lagrangian-based optimization procedure to compute the linear sensitivity of the recirculation length with respect to base flow modifications, localized bulk point forces and blowing and suction at the cylinder wall. The results are compared to nonlinear steady-state computations. The influence of a shortened or increased recirculation on the stability of the flow above threshold (Re=47) is finally discussed.

8:13AM G25.00002 Dynamic Mode Decomposition of PIV measurements for the cylinder wake flow in turbulent regime[1] , LAURENT CORDIER, GILLES TISSOT, NICOLAS BENARD, BERND R. NOACK, Institute PPRIME, France — For historical reasons, Proper Orthogonal Decomposition (POD) is the most well-known and used reduction approach in the turbulence community. POD is widely used since it extracts from a sequence of data an orthonormal basis which captures optimally the flow energy. Unfortunately, energy level is not necessarily the correct criterion in terms of dynamical modelling and deriving a dynamical system based on POD modes leads sometimes to irrelevant models. In this communication, the Dynamic Mode Decomposition (DMD) as recently proposed by Schmid (JFM 2010) will be used to determine the eigenvalues and eigenvectors of the Koopman operator (Rowley *et al.*, JFM 2009), an infinite-dimensional linear operator associated with the full nonlinear system. Without explicit knowledge of the dynamical operator, frequencies and growth rates associated to each DMD modes can be easily determined based on the eigenvalues. The DMD will be demonstrated on experimental data corresponding to a PIV dataset of a cylinder wake flow at Reynolds number 40000. Moreover, the link between DMD and temporal discrete Fourier transform will be analysed and discussed.

[1]Partially funded by the ANR Chair of Excellence TUCOROM and the ANR CORMORED

8:26AM G25.00003 Characterization of an Actively Controlled Three-Dimensional Turret Wake , PATRICK SHEA, MARK GLAUSER, Syracuse University — Three-dimensional turrets are commonly used for housing optical systems on airborne platforms. As bluff bodies, these geometries generate highly turbulent wakes that decrease the performance of the optical systems and the aircraft. The current experimental study looked to use dynamic suction in both open and closed-loop control configurations to actively control the turret wake. The flow field was characterized using dynamic pressure and stereoscopic PIV measurements in the wake of the turret. Results showed that the suction system was able to manipulate the wake region of the turret and could alter not only the spatial structure of the wake, but also the temporal behavior of the wake flow field. Closed-loop, feedback control techniques were used to determine a more optimal control input for the flow control. Similar control effects were seen for both the steady open-loop control case and the closed-loop feedback control configuration with a 45% reduction in the suction levels when comparing the closed-loop to the open-loop case. These results provide unique information regarding the development of the baseline three-dimensional wake and the wake with three different active flow control configurations.

8:39AM G25.00004 Flow Visualization of a von Kármán Ogive Forebody with Plasma Actuation , JOHN FARNSWORTH, ZACHARY FRANCIS, REID WITT, CHRIS PORTER, THOMAS MCLAUGHLIN, US Air Force Academy, USAF Academy, CO — The flow field around an axisymmetric forebody at a moderate angle of attack can produce a significant side force, and thus a yawing moment, on the body. The side force results from an asymmetric vortex state and therefore pressure distribution that forms on the body. This asymmetric vortex state originates from a convective instability in the flow field, meaning that minor geometric or flow disturbances near the apex of the model can perturb the flow into an asymmetric state. In the current experiments two single dielectric barrier discharge plasma actuators are used to perturb the flow and control the vortex state. Smoke flow visualization techniques were utilized to better understand the behavior of the vortices under plasma actuation. It was found that the vortex state responds proportionally to the voltage of the plasma actuation. Additionally, the response of the vortex state to control changed drastically with changes in Reynolds number, suggesting a relation between the blowing ratio and the behavior of the vortex state.

8:52AM G25.00005 Feedback control of vortex shedding: An explanation of the gain window[1] , SIMON ILLINGWORTH, University of Cambridge, HIROSHI NAITO, KOJI FUKAGATA, Keio University, KEIO UNIVERSITY TEAM — This presentation explains the gain window phenomenon seen in early experimental and computational studies on active, closed-loop control of vortex shedding, whereby shedding is completely suppressed only if the feedback gain lies within some narrow window of stabilizing gains. Using two-dimensional direct numerical simulations and reduced-order modeling techniques, a low-order, linear model of the cylinder wake is formed at a Reynolds number of 60. This model is used to reproduce and to explain the gain window seen in previous studies. It is shown that the gain window is not caused by the destabilization of a higher mode, but rather is determined entirely by the behaviour of the open-loop unstable mode under the action of the closed-loop controller. The time taken for actuated fluid to convect to the sensor location plays an important part in explaining this gain window. A similar analysis at a higher Reynolds number of 80 reveals that the wake remains unstable for all choices of the feedback gain. The study illustrates the limitations of closed-loop suppression of vortex shedding when a very simple control strategy is used.

[1]S.J.I. gratefully acknowledges financial support from a JSPS fellowship.

9:05AM G25.00006 A Comparative Study of Spatially Modulated Forcing of Cylinder Wake with Segmented Plasma Actuators of Different Wavelengths[1] , SAMIK BHATTACHARYA, JAMES W. GREGORY, The Ohio State University — The wake of a 1-inch diameter (D) circular cylinder was forced three dimensionally by mounting dielectric barrier discharge plasma actuators at specific spanwise locations. Actuators with spanwise wavelengths of 1, 2, 4 and 6D were used. Considerable drag reduction was achieved with 4D and 6D actuators compared to 1D and 2D actuators which showed no effect on the drag. In the case of the 6D actuator, prominent spanwise differences were observed in the wake mean velocity profiles, with a much wider wake behind the region of no plasma formation and differential development persisting well downstream. The different wavelength actuators were also compared in terms of their ability to induce streamwise vorticity in the wake. Segmented forcing with the 4D actuator augmented the generation of streamwise vorticity in the wake which was revealed in the cross-plane time-averaged data. A detailed study of vortex dislocation was carried out with a rake of eight hot wires. Segmented forcing with 4 and 6 D actuator introduced vortex dislocation in the Karman vortex street. The phenomenon of vortex dislocation due to phase mismatch is inferred as the primary reason for the effectiveness of 4D actuator in reducing drag.

[1]This work is supported by the Air Force Office of Scientific Research, Award Number FA9550-10-1-0490

9:18AM G25.00007 Open loop control of an axisymmetric turbulent wake using periodic jet blowing , JONATHAN MORRISON, ANTHONY OXLADE, Imperial College — In this study we investigate the effects of periodic 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 35%. This pressure rise is obtained only when the forcing frequency is greater than four times the frequency of the shear layer mode. The mechanism of pressure recovery is characterised via measurements of fluctuating pressure on the base of the body, and high resolution 2C PIV in the wake. Modal decomposition of the pressure fluctuations reveals a nonlinear coupling between the symmetric ($m = 0$) perturbation and higher order azimuthal modes that results in an asymmetric mean pressure distribution.

9:31AM G25.00008 Single frequency lock-on of wake behind a circular cylinder using oscillatory actuation , PHILLIP MUNDAY, KUNIHIKO TAIRA, Florida State University — We numerically investigate the influence of oscillatory flow control on the two-dimensional wake behind a circular cylinder at Re = 100. Understanding under what conditions the shedding frequency locks solely onto the actuation frequency and drag reduction is of particular interest. We investigate the influence of steady and oscillatory components of actuation with the actuator position, direction, frequency, and amplitude varied. For oscillatory forcing a V-shaped lock-on region can be seen in the frequency–amplitude plot about the natural shedding frequency, which resemble stability horns in oscillator dynamics. Steady actuation is observed to reduce mean and fluctuating components of drag. With oscillatory forcing reducing drag fluctuation allows the wake to lock-on to a wider range of actuation frequencies. We find that there exists a set of forcing amplitudes and frequencies that achieves both lock-on and drag reduction. Low frequency oscillation due to the competition between low and high drag states are observed in the vicinity of the lock-on boundary.

9:44AM G25.00009 Wake Control of a Blunt Trailing Edge Profiled Body Using Dielectric Barrier Discharge Plasma Actuators[1] , ARASH NAGHIB-LAHOUTI, PHILIPPE LAVOIE, University of Toronto Institute for Aerospace Studies — The periodic shedding of von Karman vortices is the source of cyclic aerodynamic forces on nominally 2D bluff bodies. Beyond a threshold Reynolds number, which can be as high as 700 depending on profile geometry, secondary instabilities, appearing as undulations in the von Karman vortices and pairs of counter-rotating streamwise vortices, emerge in the wake. The secondary instabilities are found to persist at Reynolds numbers in the order of 10^4. It has been shown that amplification of the secondary instabilities can lead to disorganization of the von Karman vortices, and attenuation of the cyclic forces. In the present study, this relationship is used as the basis of a wake flow control approach for a blunt trailing edge profiled body, comprised of an elliptical leading edge and a rectangular trailing edge. An array of dielectric barrier discharge plasma actuators placed at the trailing edge is used for control actuation, with a spanwise spacing based on the wavelength of the secondary instabilities, to achieve maximum amplification of the instabilities. PIV and hot-wire measurements have been conducted at Reynolds numbers between 2,000 and 24,000 to determine the effect of flow control on the wake characteristics, and the total drag.

[1]Funded in part by the Government of Ontario, Mitacs, and Bombardier Aerospace.

9:57AM G25.00010 Scaling Relations for Plasma Streamwise Vortex Generators[1] , F.O. THOMAS, M. WICKS, T.C. CORKE, University of Notre Dame, M. PATEL, Innovative Technology Applications Company, LLC. — A parametric investigation into the performance of plasma streamwise vortex generators (PSVG) for flow control was performed. The study utilized an array of PSVGs, which were flush mounted to a flat, zero pressure gradient turbulent boundary layer development plate. This work focused on characterizing the effect of freestream velocity, peak-to-peak applied voltage, inter-electrode spacing and covered electrode length on the streamwise vorticity produced by these devices. The performance of the PSVGs was also compared to that of passive vortex generators under identical flow conditions. Based upon the results of the parametric study, the flow physics of streamwise vorticity production by the PSVGs was discerned and the mechanisms are described in this paper. In addition, scaling relations are developed and presented for PSVGs, which, can be used in order to design actuator arrays for specific flow control applications.

[1]This work was supported by Innovative Technology Applications Company (ITAC), LLC under a Small Business Innovation Research (SBIR) Phase II Contract No. N00014-11-C-0267 issued by the U.S. Department of the Navy.

Monday, November 19, 2012 8:00AM - 10:10AM –

Session G26 ReactIve Flows IV: Detonations 31B - Chair: Kazhikathra Kailasanath, Naval Research Laboratory

8:00AM G26.00001 Multiplicity of detonation regimes in systems with a multi-peaked thermicity , MATEI I. RADULESCU, University of Ottawa, FAN ZHANG, Defence Research and Development Canada — Bulk exothermicity in most gaseous detonation waves occurs in a single step. There are however several physical systems displaying multiple thermicity peaks. Examples are the nuclear fusion reactions sequence in supernovae explosions, hybrid detonations in multi-phase fuels and other reactive systems. The multiplicity of steady state detonation regimes in the presence of an endothermic internal or external loss is demonstrated through analysis of the reaction zone structure described by the reactive Euler equations with two sequential Arrhenius reactions. The steady Zel'dovich - Von Neumann- Doering reaction structure is obtained numerically. The reaction zone displays embedded sonic points where the net thermicity vanishes simultaneously. Depending on the magnitude of the losses or endothermic process, the detonation wave speed response was found to have multiple steady states and turning points, which are controlled by the magnitude of the kinetic parameters of each reaction. The dependence on system parameters is established analytically using the Fickett detonation analogue model with two sequential reactions.

8:13AM G26.00002 Detonation and Transition to Detonation in Horizontal Water-Filled Pipes , NEAL P. BITTER, JOSEPH E. SHEPHERD, California Institute of Technology — Detonations and deflagration-to-detonation transition (DDT) are experimentally studied in horizontal pipes which are partially filled with water. The gas layer above the water is stoichiometric hydrogen-oxygen at 1 bar. The detonation wave produces oblique shock waves in the water, which focus at the bottom of the pipe due to the curvature of the walls. This results in peak pressures at the bottom of the pipe that are 4-6 times greater than the peak detonation pressure. Such pressure amplification is measured for water depths of 0.25, 0.5, 0.75, 0.87, and 0.92 pipe diameters. Focusing of the oblique shock waves is studied further by measuring the circumferential variation of pressure when the water depth is 0.5 pipe diameters, and reasonable agreement with theoretical modeling is found. Failure of the detonation waves was not observed, even for water depths as high as 0.92 pipe diameters. Transition to detonation also occurred at every water height, and transition distance did not vary significantly with water height.

8:26AM G26.00003 Front Structure of Three-Dimensional Detonations in Gaseous Mixtures , BOO CHEONG KHOO, National University of Singapore, HUA-SHU DOU, Zhejiang Sci-Tech University — Numerical simulations have indicated that independent of how the disturbance is imposed at the beginning of the simulations the final stable detonation in a narrow duct is always the spinning detonation with 90 degree phase difference. With a simplified model, the detonation can be described by a process of energy release with a time periodic function corresponding to the motion of transverse waves, and the energy gradient in time can be a source of instability. It is proposed that the most stable energy release is such that the time derivative of energy release is the lowest. In the unsteady detonation, all the detonation structures always tend to approach this stable state. The calculations indicate that for spinning detonation and rectangular detonation in rectangular ducts, the 90 degree phase difference of transverse waves is the most stable and the in-phase detonation is the most unstable. For oblique detonation, the 180 degree phase difference is the most stable and the in-phase detonation is the most unstable. Under a sufficiently large disturbance, the oblique detonation can finally involve into rectangular mode with 90 degree phase difference. These results are in agreement with the numerical simulations and experiments.

8:39AM G26.00004 Pressure Feedback in Rotating Detonation Engines[1] , DOUGLAS SCHWER, K. KAILASANATH, Naval Research Laboratory — Rotating detonation engines (RDEs) represent a unique method for obtaining propulsion from the high efficiency detonation cycle. In order for the RDE to be a practical propulsive device, engines must be capable of running efficiently at low pressure ratios, however, this type of injection typically results in a large amount of pressure feedback into the injection system. This paper examines different aspects of the pressure feedback phenomena, and investigates approaches to injecting fresh mixture that reduce the amount of feedback.

[1]This work is supported by the Office of Naval Research through NRL 6.1 Computational Physics Task Area.

8:52AM G26.00005 The exhaust flow field of a rotating detonation-wave engine[1] , KAZHIKATHRA KAILASANATH, DOUGLAS SCHWER, U.S. Naval Research Laboratory — Rotating detonation-wave engines (RDE) are a form of continuous detonation-wave engine. They potentially provide further gains than an intermittent or pulsed detonation–wave engine (PDE). However, significantly less work has been done on this concept when compared to the PDE. Last year, we presented the details of the injection system on the overall flow field in an RDE. In this talk, we focus on the effects of adding an exhaust plenum to this idealized RDE. While the overall exhaust flow shows that a recirculation zone sets up behind the RDE when a plenum is added, the net effect on the flow field within the RDE and on performance is found to be small. However, the slight modification to the flow field may impact the design of suitable nozzles for this device. This is explored further with the addition of a simple conical nozzle. This nozzle reduces the size of the recirculation zone and also reduces the temperature in the plume but has little effect on the flow field inside the RDE.

[1]Work Sponsored by ONR through NRL 6.1 Computational Physics Task Area

9:05AM G26.00006 Capturing the Dynamics of Unsteady Inviscid and Viscous Hydrogen-Air Detonations , CHRISTOPHER ROMICK[1], University of Notre Dame, TARIQ ASLAM[2], Los Alamos National Laboratory, JOSEPH POWERS[3], University of Notre Dame — We consider the calculation of one-dimensional unsteady detonation in a mixture of calorically imperfect ideal gases with detailed kinetics. Both inviscid and viscous detonations of an initially stoichiometric hydrogen-air mixture at ambient conditions of $293.15\ K$ and $0.421\ atm$ are considered using a chemical mechanism composed of 19 reversible reactions, containing 9 species and 3 elements. The use of detailed kinetics introduces multiple reaction length scales, and their interaction gives rise to complex dynamics. In the inviscid limit, both shock-capturing and shock-fitting are used on a uniform grid. The diffusive behavior is predicted using a wavelet-based adaptive mesh refinement technique and includes multi-component species, momentum, and energy diffusion, as well as DuFour and Soret effects. In the inviscid limit when using shock-capturing, finer resolutions are necessary to accurately capture the dynamics in the unstable regime than when using shock-fitting. At the resolutions necessary for accurate shock-capturing, diffusion can play a crucial role in determining the overall behavior. Near the neutral stability point, the addition of physical diffusions dampens the amplitude of oscillations significantly.

[1]Department of Mechanical & Aerospace Engineering
[2]Weapons Experiments Division
[3]Department of Mechanical & Aerospace Engineering

9:18AM G26.00007 Developing Subgrid Models for Shock-to-Detonation Mesoscale Simulations , THOMAS JACKSON, University of Illinois at Urbana-Champaign — Determining the thermal and mechanical sensitivity of new and existing energetic materials is important for transportation, safety and storage concerns. Initiation of an energetic material can occur when an impulse delivered to the material evolves into a self-sustaining detonation wave. The microstructure can lead to local regions of high temperature, so-called "hot spots." Temperatures in these hot spots can exceed the bulk temperature expected from shock heating, which in turn can trigger ignition even when a homogenized model might fail to predict it. If the chemical and mechanical energy release within hot spots exceeds cooling by diffusion and join up, a localized ignition can occur. Ignition spread due to evolution and growth of high-temperature regions, potentially with reinforcement from neighboring regions or preconditioning of the material, can then lead to detonation. Hot spots are thought to be formed due to shock interaction with microscale and molecular-scale material inhomogeneities through processes such as void collapse, shear banding, debonding, and grain sliding. The most important question at the device scale is whether or not the individual hot spots will coalesce to create a local ignition front, and whether the ignition front or fronts are in turn sufficient to initiate the entire device. Our approach has two principal steps. We first develop sub-grid models based on hot-spot dynamics, and then use the sub-grid model in our mesoscale simulations using our shock dynamics code. In this talk we present recent efforts into developing subgrid models that can be incorporated into mesoscale simulation codes.

9:31AM G26.00008 A model for shock wave chaos[1] , LUIZ FARIA, ASLAN KASIMOV, KAUST, RODOLFO ROSALES, MIT — We propose the following simple model equation that describes chaotic shock waves:

$$u_t + \frac{1}{2}\left(u^2 - uu_s\right)_x = f\left(x, u_s\right).$$

It is given on the half-line $x < 0$ and the shock is located at $x = 0$ for any $t \geq 0$. Here $u_s\left(t\right)$ is the shock state and f is a given source term [1]. The equation is a modification of the Burgers equation that includes non-locality via the presence of the shock-state value of the solution in the equation itself. The model predicts steady-state solutions, their instability through a Hopf bifurcation, and a sequence of period-doubling bifurcations leading to chaos. This dynamics is similar to that observed in the one-dimensional reactive Euler equations that describe detonations. We present nonlinear numerical simulations as well as a complete linear stability theory for the equation.

[1]Supported by DMS-0907955 and KAUST Office of Competitive Research Grants.

9:44AM G26.00009 On gaseous detonation in a radially expanding flow , ASLAN KASIMOV, SVYATOSLAV KORNEEV, KAUST — We investigate two-dimentional converging detonation in a radially expanding flow of ideal gas. The steady state structure is computed analytically and its stability and nonlinear dynamics are explored using numerical simulation. Intricate cellular patterns are observed.

9:57AM G26.00010 Numerical Simulations of Detonation Wave - Magnetic Field Interactions[1] , LORD COLE, ANN KARAGOZIAN, University of California, Los Angeles — Numerical simulations of one- and two-dimensional detonation waves subjected to an applied magnetic field are performed, with applications to flow control and MHD thrust augmentation in Pulse Detonation Engines and their design variations.[2] The evolution of the ionization processes and the diffusive and convective transport of the magnetic field are examined in the context of their effect on detonation dynamics. As with prior studies on hydrogen-air detonation dynamics,[3] the present studies explore hydrogen-air-cesium detonations via high order shock capturing schemes and complex reaction kinetics, in addition to a two-temperature relaxation model for the plasma. One-dimensional simulations examining the non-coupled effect of the magnetic field on the unsteady detonation indicate that the stabilizing effect of the dilluent, cesium, becomes less effective when it becomes an active participant under the influence of strong magnetic fields. Two-dimensional dynamics allow a more complete coupling between the magnetic field and the detonation kinetics to be represented, with implications for an alteration in stability characteristics.

[1]Supported by the US Air Force/ERC, Inc. under subcontract RS100226.
[2]Zeineh, et al., **J. Prop. & Power**, Vol. 28, No. 1, pp. 146-159, 2012
[3]Cole, et al., **Comb. Sci. & Tech.**, to appear, 2012

Monday, November 19, 2012 8:00AM - 10:10AM –

Session G27 Separated Flows II 31C - Chair: Manoochehr Koochesfahani, Michigan State University

8:00AM G27.00001 Shock wave unsteadiness in an over-expanded nozzle , BRITTON OLSON, SANJIVA LELE, Stanford University — A suite of Large-Eddy Simulations has recently elucidated the unsteadiness of a shock wave in an over-expanded planar nozzle. The simulations model the nozzle used by Johnson and Papamoschou (*P*hys. Fluids **22**, 2010), who found that the exhaust jet was destabilized by the shock wave oscillations. Shock wave unsteadiness has been observed in several experiments with similar nozzle geometries. The mechanism which drives the instability is a feedback loop between the nozzle exit and the shock wave. The shock boundary layer interaction causes flow separation and reversal, which then causes an obstruction at the exit of the nozzle. The obstruction is seen as a change in the effective exit area, which in turn causes the shock to readjust its position. When the shock moves, the nature of the shock induced separation changes and the cycle repeats, never becoming stationary. Parametric variation of the nozzle geometry and pressure ratio demonstrate that the instability has a dependence on the Mach number and Reynolds number. A reduced order model (ROM) which is based on the proposed mechanism and the LES data is developed. Preliminary results indicate that the ROM predicts the frequency of the instability to within 10% when compared to the LES data.

8:13AM G27.00002 Sensitivity of an asymmetric three-dimensional diffuser to inlet condition perturbations[1] , EMILY SAYLES, Stanford University, SVEN GRUNDMANN, TU Darmstadt, CHRISTOPHER ELKINS, JOHN EATON, Stanford University — Experiments were performed to investigate the flow in an asymmetric 3D diffuser that is highly sensitive to inlet condition perturbations. Magnetic Resonance Velocimetry (MRV) revealed the development of a three-dimensional separation bubble in the baseline case. The shape and size of this separation bubble could be manipulated through the introduction of longitudinal counter-rotating vortices produced by small delta-wing vortex generators placed in the inlet duct. The changes to the separation bubble were reflected in significant changes in the diffuser's pressure recovery. Similar pressure recovery effects were observed by perturbing the inlet flow with dielectric barrier discharge plasma actuators oriented to generate forcing in the spanwise direction. The plasma actuators can both improve and degrade the diffuser's performance. Two cases, a continuous forcing case which decreases the pressure recovery and a pulsed forcing case which increases the pressure recovery, were selected for further study. Particle image velocimetry was used to better understand how the secondary flows introduced by the plasma actuators interact with the separation bubble, and why they have such a marked effect on the diffuser's performance.

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

8:26AM G27.00003 Triple deck solutions for supersonic flow past obstacles , RAMESH YAPALPARVI, Florida State University, PIERRE-YVES LAGREE, Institut Jean Le Rond d'Alembert — This study is based on the numerical investigation of the boundary-layer separation of supersonic flow at high Reynolds number past obstacles (flat plate with a hump) based on the concept of viscous-inviscid interaction. The "triple-deck" model is used to describe the interaction process. We observe a separation region both ahead and behind the hump whereas a separation region is centered for the indents. In case of humps, at higher values of hump height, the pressure has both positive and negative value of "plateau" (observed up to now only upstream wedges). This novel feature has been compared with the "interacting boundary-layer" model which shows an excellent agreement.

8:39AM G27.00004 An experimental study of flow past a rotationally oscillating cylinder. , SANJAY KUMAR, CARLOS LOPEZ, The University of Texas at Brownsville, OLIVER PROBST, Instituto Tecnologico y de Estudios Superiores de Monterry, Mexico, DAVOOD ASKARI, YINGCHEN YANG, The University of Texas at Brownsville — Flow past a circular cylinder executing sinusoidal rotary oscillations about its own axis is studied experimentally. The experiments are carried out at Re = 185, oscillation amplitudes varying from $\pi/8$ to π, and forcing frequency ratios varying from 0 to 5. It is found that the phenomenon of lock-on occurs in a forcing frequency range which depends not only on the oscillation amplitude but also the downstream location from the cylinder. The experimentally measured lock-on diagram in the forcing amplitude and frequency plane is presented at various downstream locations ranging from 2 to 23 diameters. The upper limit of the lock-on forcing frequency band depends strongly on the downstream location whereas the lower limit is fairly insensitive. The far field wake decouples, after the lock-on at higher forcing frequencies and behaves more like a regular Karman vortex street from a stationary cylinder with a vortex shedding frequency mostly lower than the one from a stationary cylinder. The dependence of circulation values of shed vortices on the forcing frequency revealed a universal decay curve independent of forcing amplitude beyond forcing frequency of ~ 1.0.

8:52AM G27.00005 Shear-Layer Interactions Between Surface-Mounted Obstacles at Varying Streamwise Spacings , T. KIM, J.L. BEST, K.T. CHRISTENSEN, University of Illinois at Urbana-Champaign — Surface obstacles occur in a variety of flows, such as roughness elements in engineering flows and barchan dunes in natural eolian environments on both the Earth and Mars. Depending upon the arrangement and spacing between such obstacles, the flow over one obstacle can significantly alter the flow over those positioned downstream. Such flow interactions occur in fields of barchan dunes that are closely spaced and aligned in the flow direction, and where flow sheltering may play a significant role. To better understand these flow interactions, experiments were conducted for a pair of identical, upright cylinders extending into the log layer and aligned at various spacings in the streamwise direction of a turbulent channel flow at $\mathrm{Re}_\tau \sim 1200$. Particle-image velocimetry measurements of the flow around the cylinders reveal strong interactions between the shear layers generated downstream of the cylinders, and particularly a weakening of the downstream-most shear layer for small cylinder spacings ($< 4 - 6D$). Modifications of the vortex-shedding processes at the downstream cylinder are under investigation, as these interactions are thought to play a critical role in the formation and evolution of surface obstacles when the surface is cohesionless and mobile.

9:05AM G27.00006 Generalised phase average with applications to sensor-based flow estimation of the wall-mounted square cylinder wake[1] , ROBERT MARTINUZZI, University of Calgary, JASON BOURGEOIS, Unversity of Calgary, BERND R. NOACK, Institute PPRIME, France — We experimentally investigate the three-dimensional wake behind a finite wall-mounted square cylinder at $Re = 12,000$ and aspect ratio of 4. Focus is placed on the base flow and oscillatory fluctuation. Time-resolved 3D velocity fields are constructed from high-frame-rate particle image velocimetry (PIV) and simulatenously recorded surface pressure measurements. All three velocity components are resolved in a rectangular near-wake region by two orthogonal dense arrays of parallel PIV planes. A key enabler is a generalized phase-average incorporating slowly varying base flow, a variable oscillation amplitude and higher harmonics. These generalizations reduce the residual 30% below those of a traditional phase average. Moreover, the resolved variations reveal analytical constraints of the mean flow and oscillation levels, like the mean-field paraboloid. The proposed methodology for generalized phase averaging and for construction of 3D velocity fields from 2D PIV data is applicable to a large class of turbulent flows with oscillatory dynamics.

[1]Partially funded by the NSERC Discovery programme and ANR Chair of Excellence TUCOROM.

9:18AM G27.00007 Low-Reynolds-number vortex dynamics around moving wings , RYAN JANTZEN, KUNIHIKO TAIRA, Florida State University, MICHAEL OL, KENNETH GRANLUND, U.S. Air Force Research Lab — The focus for the present research is to investigate the fundamental flow physics around low-aspect-ratio flat-plate wings undergoing pitching and surging motions. Numerical simulations performed with the immersed boundary projection method are used to investigate the three-dimensionality of the low-Reynolds-number flow around these wings. Of particular interest is the influence of wing motion on the formation of the leading-edge, trailing-edge, and tip vortices. To determine the relationship that these pure motions have on the formation of these vortices, we vary the aspect ratio, pitching rate, and pivot-point location. The spanwise variation in the roll-up of the leading and trailing-edge vortices under the influence of tip effects is analyzed. The aerodynamic forces generated during these unsteady wing motions will be compared to force measurements obtained from moderate-Reynolds-number towing tank experiments. A further understanding of the underlying flow physics for these idealized motions is necessary in order to understand more complex wing maneuvers.

9:31AM G27.00008 Wake Modes and Heat Transfer from Rotationally Oscillating Cylinder , PRABU SELLAPPAN, TAIT POTTEBAUM, University of Southern California — Wake formation is an important problem in engineering due to its effect on phenomena such as vortex induced vibrations and heat transfer. While prior work has focused on the wake formation due to vortex shedding from stationary and oscillating cylinders, limited information is available on the relationship between wake modes and heat transfer from rotationally oscillating cylinders. Experiments were carried out at Re=150 and 750, using an electrically heated cylinder, in a water tunnel for oscillation frequencies from 0.67 to 3.5 times the natural shedding frequency and peak-to-peak oscillation amplitudes up to 320°. DPIV was used to identify and map wake modes to various regions of the parameter space. Temperature data from a thermocouple embedded in the cylinder was used to calculate heat transfer rates. Correlation between heat transfer enhancement and certain wake mode regions were observed in the parameter space. The relationship between wake formation and heat transfer enhancement will be described.

9:44AM G27.00009 Effect of approach flow on the bluff body wake behind a ship superstructure[1] , CODY BROWNELL, LUKSA LUZNIK, HYUNG SUK KANG, MURRAY SNYDER, United States Naval Academy — Air velocity measurements are obtained in situ aboard a 108 ft naval training vessel operating in the Chesapeake Bay. Three-component sonic anemometers are placed on a vertical mast at the bow of the ship, for approach flow measurement, and at numerous locations above a flight deck at the stern of the ship. The mean flow structure resembles that of a 3D backward-facing step, with a recirculation region covering much of the flight deck. The flow at the bow, mostly undisturbed by the presence of the ship, is characterized in the region up to 11-m above the sea surface. The effects of approach angle and atmospheric stability on the wake turbulence is discussed.

[1]Supported by ONR

9:57AM G27.00010 Reducing the Drag and Damage of a High-Speed Train by Analyzing and Optimizing its Boundary Layer Separation and Roll-up into Wake Vortices , CHUNG-HSIANG JIANG, PHILIP MARCUS, UC Berkeley — We present numerical calculations of the boundary layers and shed wake vortices behind several aerodynamic bodies and generic models of high-speed trains. Our calculations illustrate new visual diagnostics that we developed that clearly show where the separation of a boundary layer occurs and where, how, and with what angles (with respect to the stream-wise direction) the wake vortices form. The calculations also illustrate novel 3D morphing and mesh "pushing and pulling" techniques that allow us to change the shapes of aerodynamic bodies and models in a controlled and automated manner without spurious features appearing. Using these tools we have examined the patterns of the shed vortices behind generic bodies and trains and correlated them with the changes in the drag as well as with the effects of the shed vortices on the environment. In particular, we have applied these techniques to the end car of a next-generation, high-speed train in order to minimize the drag and to minimize the adverse effects of the shed vortices on the track ballast.

Monday, November 19, 2012 8:00AM - 10:10AM –

Session G28 Waves I 32A - Chair: Kyle Brucker, SAIC

8:00AM G28.00001 The Influence of surface waves on marine current turbine performance[1] , ETHAN LUST, KAREN FLACK, LUKSA LUZNIK, United States Naval Academy — Performance characteristics are presented for a $1/25^{th}$ 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 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 a depth of 0.8D measured from the blade tip to the mean free surface. The performance characteristics without waves match expected results from blade-element-momentum theory. Results show that the average power coefficient is unaffected by the presence of waves, however, the phase averaged results indicate significant variation with wave phase.

[1]Work supported by ONR.

8:13AM G28.00002 Laboratory study of the structure of the airflow and separation above surface waves , FABRICE VERON, MARC BUCKLEY, University of Delaware, School of Marine Science and Policy — The effects of the surface waves on the airflow dynamics greatly influence the flux of momentum between the ocean and the atmosphere. While we know that most the surface stress is supported by the wave-coherent stress carried in large part by the small gravity-capillary wind waves, the role of the airflow separation above these small waves is not well understood. 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 velocities were obtained using high resolution concatenated PIV, and wave profiles and spectra were measured by dual-beam laser-induced fluorescence. We observe direct evidence of intermittent airflow separation 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, proper orthogonal decomposition (POD) of the wave phase-locked velocity products suggests that airflow separation generates intense mixing and transport of surface generated vorticity within the airflow. These results hold for wind speeds that would normally be considered low to moderate.

8:26AM G28.00003 Impact of Plunging Breaking Wave on a Partially Submerged Cube[1] , A. WANG, C.M. IKEDA, J.H. DUNCAN, University of Maryland — The impact of a plunging breaking wave on a partially submerged rigid cube ($L = 30.5$ cm) is studied experimentally in a wave tank that is 14.8 m long, 1.15 m wide and 2.2 m high with a water depth of 0.91 m. A single repeatable plunging breaker is generated from a dispersively focused wave packet (average frequency of 1.14 Hz) that is created with a programmable wave maker. The water surface profiles at the vertical center plane of the cube are measured with a cinematic LIF technique. The cube is centered in the width of the tank and mounted from above with the front face oriented with its normal in the vertical long center plane of the tank and tilted at angles of 0 and 20 degrees downward relative to horizontal. For the range of horizontal cube positions used here, during the wave impact, the water free surface forms a circular arc between the water contact point on the front face of the cube and the wave crest. As the wave impact continues, this arc converges to a point and a fast-moving vertical jet is formed. The effect of the submergence and tilt angle of the cube on the jet formation are explored.

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

8:39AM G28.00004 Crosswaves induced by the vertical oscillation of a fully immersed vertical plate , FREDERIC MOISY, Laboratoire FAST, Universite Paris-Sud, GUY-JEAN MICHON, Laboratoire FAST, UPMC, MARC RABAUD, Laboratoire FAST, Universite Paris-Sud, ERIC SULTAN, Laboratoire FAST, UPMC — Capillary waves excited by the vertical oscillation of a thin elongated plate below an air-water interface are analyzed using time-resolved measurements of the surface topography. A parametric instability is observed above a well defined acceleration threshold, resulting in a so-called cross-wave, a staggered wave pattern localized near the wavemaker and oscillating at half the forcing frequency. This cross-wave, which is stationary along the wavemaker but propagative away from it, is described as the superposition of two almost anti-parallel propagating parametric waves making a small angle of the order of 20 degres with the wavemaker edge. This contrasts with the classical Faraday parametric waves, which are exactly stationary because of the homogeneity of the forcing. Our observations suggest that the selection of the cross-wave angle results from a resonant mechanism between the two parametric waves and a characteristic length of the surface deformation above the wavemaker.

8:52AM G28.00005 Numerical Simulation of a Seaway with Breaking , DOUGLAS DOMMERMUTH, THOMAS O'SHEA, KYLE BRUCKER, DONALD WYATT, Naval Hydrodynamics Division, Science Applications International Corporation — The focus of this presentation is to describe the recent efforts to simulate a fully non-linear seaway with breaking by using a high-order spectral (HOS) solution of the free-surface boundary value problem to drive a three-dimensional Volume of Fluid (VOF) solution. Historically, the two main types of simulations to simulate free-surface flows are the boundary integral equations method (BIEM) and high-order spectral (HOS) methods. BIEM calculations fail at the point at which the surface impacts upon itself, if not sooner, and HOS methods can only simulate a single valued free-surface. Both also employ a single-phase approximation in which the effects of the air on the water are neglected. Due to these limitations they are unable to simulate breaking waves and air entrainment. The Volume of Fluid (VOF) method on the other hand is suitable for modeling breaking waves and air entrainment. However it is computationally intractable to generate a realistic non-linear sea-state. Here, we use the HOS solution to quickly drive, or nudge, the VOF solution into a non-linear state. The computational strategies, mathematical formulation, and numerical implementation will be discussed. The results of the VOF simulation of a seaway with breaking will also be presented, and compared to the single phase, single valued HOS results.

9:05AM G28.00006 PIV measurements of the interaction between a surface wave and a moving sphere , ANDREW RYDALCH, ERIC MAXEINER, IVAN SAVELYEV, None — The objective of this experiment was to study the interaction between the wake of a moving sphere and a surface wave moving in the same direction. The tests were conducted in an 8.5 x 2.8 m freshwater tank in 30cm of water. An underwater tow track, consisting of a belt-driven carriage riding on aluminum rails mounted on the bottom of the tank, was constructed in order to leave the free surface of the water unperturbed to allow for precise measurements at the air-water interface. In addition to the towing track, the tank also contains a single paddle wave-maker. PIV measurements were taken of the wake as the sphere moved in conjunction with the wave packet. The sphere velocity was constant and the motion was synchronized with the wave maker such that the sphere traveled inside the crest of a wave as it passed through the test area. In order to identify disturbances to the mean flow caused by the wave, the results were compared with measurements of the wake of the same sphere moving through quiescent water, and also of the wave field without the sphere present. In both comparisons, the measurements can be subtracted from the wave/sphere case to characterize the interaction. The results are applicable to motion control systems for vehicles operating in the presence of surface waves.

9:18AM G28.00007 On the unsteady free surface wave pattern found behind a localized pressure distribution moving at speeds just below the minimum phase speed of linear gravity capillary waves[1] , N. MASNADI, J.H. DUNCAN, University of Maryland — The non-linear response of a water free surface to a localized pressure distribution moving at constant speed just below the minimum phase speed ($C_{min} \approx 23$ cm/s) of gravity-capillary waves is studied experimentally in a long tank. The pressure distribution is generated by blowing air onto the water surface via a vertically oriented 2-mm-ID tube that is mounted on an instrument carriage that is in turn set to move along the tank at constant speeds between 20 and 23 cm/s. A cinematic light refraction method is used to obtain quantitative measurements of the surface deformation pattern behind the air jet. At towing speeds just below C_{min}, an unsteady V-shaped wave pattern appears behind the pressure source. From observations of the wave pattern evolution, it is found that localized depressions are generated near the pressure source and propagate in pairs along the two arms of the V-shaped pattern. These are eventually shed from the tips of the pattern and rapidly decay. Measurements of the evolution of the speed of these localized depression patterns are compared to existing measurements of the speeds of steady three-dimensional solitary gravity-capillary waves (lumps) that appear behind the pressure source at even lower towing speeds.

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

9:31AM G28.00008 Spatial focusing and breaking of surface waves[1] , GERARDO RUIZ-CHAVARRIA, Facultad de Ciencias, Universidad Nacional Autonoma de Mexico, PATRICE LE GAL, MICHAEL LE BARS, IRPHE -UMR 7342 — In this work, we present experimental results about the evolution of a monochromatic wave generated with a parabolic wavemaker. Experiments were carried out in the deep water limit, with frequencies ranging between 3 and 10 Hz. Measurement of surface deformation during the wave breaking was made with a laser sheet that illuminates the interface on the symmetry axis. On the other hand for small amplitude deformation we use the Schlieren method which allows a reconstruction of the fluid surface in a two dimensional domain. The spatial focusing produces a growth of the wave amplitude and the maximal value (if no breaking occurs) is attained after the wave crosses the caustic. Otherwise, wave breaking appears initially at the axis of symmetry, before the origin of the cusp. Some energy is dissipated along the wave propagation and focusing. The wave breaking is followed by a decrease in the amplitude, but in some cases the focusing stops this process. Then the amplitude grows again and another wave breaking can be observed. Our results are compared with the prediction of linear wave theory and with numerical simulation of surface non linear waves. Finally we present a discussion of the evolution of the wave before and after the breaking.

[1]G.Ruiz-Chavarria acknowledges support by DGAPA-UNAM under project IN116312

9:44AM G28.00009 Mechanisms for wave generation in a turbulent air-water flow , FRANCESCO ZONTA, MIGUEL ONORATO, Dept. Physics, University of Torino, ALFREDO SOLDATI, Center for Fluid Mechanics and Hydraulics, University of Udine — Momentum and scalar transport phenomena across an air-water interface are important in many geophysical processes (absorption of CO_2 by the ocean) and industrial applications (dynamics of steam-air mixtures flowing into nuclear reactors). We used Direct Numerical Simulations (DNS) to explore the dynamics of countercurrent air/water flow. The motion of the air/water interface was computed by solving an advection equation for the interface vertical elevation (boundary fitted method). At each time step, the physical domain was mapped into a rectangular domain using a nonorthogonal transformation. Continuity and Navier-Stokes equations were first solved separately in each domain, then coupled (velocity/stress) at the interface. We performed DNS in the Weber, Froude and Reynolds number (We,Fr,Re) parameter space. Depending on We, Fr and Re, we obtained different transients for wave evolutions, from small capillary waves to longer gravity waves. For steady-state conditions, we observed that interface deformation echanced turbulence activity/transfer mechanisms across the interface.

9:57AM G28.00010 Observation of star-shaped surface gravity waves , JEAN RAJCHENBACH, Laboratoire de Physique de la Matiere Condensé CNRS UMR 7336, Université de Nice - Sophia Antipolis, Nice, DIDIER CLAMOND, Laboratoire J.-A. Dieudonné CNRS UMR 7351, Université de Nice - Sophia Antipolis, Nice, ALPHONSE LEROUX, Laboratoire de Physique de la Matiere Condensée CNRS UMR 7336, Université de Nice - Sophia Antipolis, Nice — We report a new type of standing gravity waves of large amplitude, having alternatively the shape of a star and of a polygon. This wave is observed by means of a laboratory experiment by vibrating vertically a tank. The symmetry of the star (i.e. the number of branches) is independent of the container form and size, and can be changed according to the amplitude and frequency of the vibration. We show that this wave geometry results from nonlinear resonant couplings between three waves, although this possibility was denied for pure gravity waves up to now.

Monday, November 19, 2012 8:00AM - 9:44AM –

Session G29 Porous Media Flows IV 32B - Chair: Emmanuel Villermaux, Aix Marseille University

8:00AM G29.00001 Self-similarity in coupled Brinkman\Navier-Stokes flows[1] , ILENIA BATTIATO, Clemson University — Coupled flows through and over porous layers occur in a variety of natural phenomena, biological systems and industrial processes. In this work we derive self-similar solutions of flows through both a porous medium and a pure fluid. Self-similar filtration velocity and hydrodynamic shear profiles are obtained by means of asymptotic analysis in the limit of infinitely small permeability, and for both laminar and turbulent regimes over the porous medium. We show that a spatial length scale, related to the porous layer thickness, naturally emerges from the limiting process and suggests a more formal definition of thin and thick porous media. The results of the analysis are applied to porous media constituted of patterned cylindrical obstacles, which can freely deflect under the shear exerted by the fluid flowing through and over the forest. A self-similar solution for the bending profile of the elastic cylindrical obstacles is obtained as intermediate asymptotic, and applied to carbon nanotube (CNT) forests' response to aerodynamic stresses. This self-similar solution is successfully used to estimate flexural rigidity of CNTs by linear fit of appropriately rescaled maximum deflection and average velocity measurements.

[1]Funding from BP International within the ExploRe program is gratefully acknowledged.

8:13AM G29.00002 Application of method of volume averaging coupled with time resolved PIV to determine transport characteristics of turbulent flows in porous bed[1] , VISHAL PATIL, JAMES LIBURDY, Oregon State University — Turbulent porous media flows are encountered in catalytic bed reactors and heat exchangers. Dispersion and mixing properties of these flows play an essential role in efficiency and performance. In an effort to understand these flows, pore scale time resolved PIV measurements in a refractive index matched porous bed were made. Pore Reynolds numbers, based on hydraulic diameter and pore average velocity, were varied from 400-4000. Jet-like flows and recirculation regions associated with large scale structures were found to exist. Coherent vortical structures which convect at approximately 0.8 times the pore average velocity were identified. These different flow regions exhibited different turbulent characteristics and hence contributed unequally to global transport properties of the bed. The heterogeneity present within a pore and also from pore to pore can be accounted for in estimating transport properties using the method of volume averaging. Eddy viscosity maps and mean velocity field maps, both obtained from PIV measurements, along with the method of volume averaging were used to predict the dispersion tensor versus Reynolds number. Asymptotic values of dispersion compare well to existing correlations. The role of molecular diffusion was explored by varying the Schmidt number and molecular diffusion was found to play an important role in tracer transport, especially in recirculation regions.

[1]Funding by NSF grant 0933857, Particulate and Multiphase Processing.

8:26AM G29.00003 Dispersive and mixing characteristics for turbulent porous media flows based on local length and time scale measurements[1] , JAMES LIBURDY, VISHAL PATIL, Oregon State University — Porous media flows have a very wide range of applications, both in engineering applications and natural flows. Local mixing and dispersion is strongly influenced by the complex pore geometry. Understanding mixing properties requires knowledge of the range of scales present within the flow and how they vary with Reynolds number. Experiments have been conducted using time resolved two component PIV based on refractive index matching of the solid and liquid phases. The flow characteristics vary over a large range of Reynolds numbers, typically based on an average pore velocity and hydraulic diameter or bead size as the characteristic length. In this study we examine the effect of increased pore Reynolds number on the turbulence characteristics for Reynolds numbers from approximately 400 to 4000. In particular the integral and Kolmogorov length scales are estimated, along with the determination of the integral velocity and Eulerian time scales. These are then used to estimate the Lagrangian time scale. The asymptotic behavior associated with increasing pore Reynolds number is shown, and used to evaluate the scaling relationships. Results are also used to demonstrate the evaluation of the mechanical dispersion coefficient and that it compares well with results obtained using global methods such as solute breakthrough curves.

[1]Funding by NSF grant 0933857, Particulate and Multiphase Processing.

8:39AM G29.00004 DNS of turbulent flow in a porous unit cell[1] , SOURABH APTE, JUSTIN FINN, BRIAN WOOD, JAMES LIBURDY, Oregon State University — Turbulent flows through packed beds and porous media are encountered in a number of natural and engineered systems, however our general understanding of moderate and high Reynolds number flows is limited to mostly empirical and macroscale relationships. In this work the porescale flow physics, which are important to properties such as bulk mixing performance and permeability, are investigated using Direct Numeric Simulation of flow through a periodic face centered cubic (FCC) unit cell. This low porosity arrangement of spheres is characterized by rapid flow expansions and contractions, and thus features an early onset to turbulence [Hill & Koch, JFM 2002]. The simulations are performed using a fictitious domain approach [Apte et al, J. Comp. Physics 2009], which uses non-body conformal Cartesian grids, with resolution up to $D/\Delta = 250$ (354^3 cells total). Simulations are performed at three pore Reynolds numbers, $Re_p = 300,\ 550$ and 950, spanning a broad physical regime. The results are used to investigate the structure of turbulence in the Eulerian and Lagrangian frames, the distribution and budget of turbulent kinetic energy, and the characteristics of the energy spectrum in complex packed beds and porous media.

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

8:52AM G29.00005 Transport in Porous Fins From Laminar to Turbulent Regime , FILIPPO COLETTI, Stanford University, KENSHIRO MURAMATSU, DENSO Corporation, BRIAN FURCINITI, CHRIS ELKINS, JOHN EATON, Stanford University — Lotus type porous metal has elongated pores of random size and spatial distribution but a common orientation. Sets of so-called lotus fins are obtained by slicing the metal into thin layers and stacking them in the flow path, forcing the fluid to pass through the pores. Lotus fins represent a promising alternative to metal foam heat exchangers, because they offer higher thermal conductivity and lower pressure drop. We have experimentally analyzed the fluid flow and heat transfer in lotus fins to determine their transport properties in a range of flow regimes. The investigated Reynolds numbers based on the pore diameter and inner velocity ranged from 100 to 4000. Three-dimensional mean velocity fields were obtained by magnetic resonance velocimetry performed on magnified replicas of the fins, allowing determination of the mechanical dispersion imposed by the random structure of the fins. Thermal measurements on non-conductive fins provided the global diffusivity coefficient, which accounts for molecular, mechanical and (at high Reynolds number) turbulent diffusion. The latter contribution was isolated and its relevance assessed as a function of the flow regime.

9:05AM G29.00006 Instability onset and mixing by diffusive Rayleigh-Benard Convection in a Hele-Shaw Cell , DANA EHYAEI, KEN KIGER, University of Maryland — The injection and eventual dissolution of carbon dioxide in deep saline aquifers has suggested as an effective means of carbon sequestration. Typical injection conditions produce a buoyantly stable source of CO2 layered on top of the brine, whose dissolution is greatly accelerated by the onset of dissolution-driven, negatively buoyant, convective plumes that develop at the interface. The current work is a study conducted within a Hele-shaw cell, as an analogue for porous media, using working fluids that are mixtures of methanol and ethylene glycol diffusing in water, imitating the convective behavior of CO2 in the brine. The underlying physics of the flow are examined by measuring the velocity field directly via PIV, using appropriate methods to allow quantitative measurement in this thin-gap flow. This technique allows for detailed measurement of the entire evolution of the velocity and vorticity field during onset, growth and saturation of the instabilities. Features of the flow, the mechanisms that govern it and accurate time scales form onset time to later time mixings would be discussed for different Rayleigh numbers ranging from 2000 to 15000.

9:18AM G29.00007 Mixing properties of stationary flows in porous media , MIHKEL KREE, EMMANUEL VILLERMAUX, Aix Marseille University — The interplay between stretching of fluid particles and molecular diffusion leads to enhanced mixing of scalar concentration fields, like in random, turbulent flows. Similarly, the flow in a porous medium develops high strain rates due to the no-slip boundary condition at solid surfaces, altering substantially molecular mixing. We report here on experiments of mixing by a stationary flows in a three-dimensional random stack of solid spheres. Two distinctive fluorescent dyes (with concentrations C_1 and C_2) are injected from separate sources and their evolution through the medium is directly observed, this being made possible by matching the refractive indices of the spheres and of the flowing liquid. We quantify the dispersion, concentration distributions, and correlation between the two fields as a function of downstream distance. The value of initially negative correlation coefficient asymptotically reaches zero, meaning that the overall field $C_1 + C_2$ is a random superposition of the individual fields. The relevant time (distances) scales for mixing are identified.

9:31AM G29.00008 Macroscopic model of unstable two-phase flow in a Hele-Shaw cell , LUIS CUETO-FELGUEROSO, RUBEN JUANES, Massachusetts Institute of Technology — When a less viscous fluid displaces a more viscous one in the gap between two parallel plates (a Hele-Shaw cell) or in a porous medium, the displacement front is unstable, and the hydrodynamic instability that ensues is referred to as viscous fingering. The emerging 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, the injection rate and gap size, and the wetting properties of the system. Here we present a continuum model of two-phase flow in Hele-Shaw cells that reproduces the observed displacement patterns. The key feature of our model is that it captures the dynamic distribution of fluids in the gap, in the sense that the thickness of the film of more viscous fluid left attached to the wall depends on the capillary number.

Monday, November 19, 2012 8:00AM - 9:57AM –

Session G30 Nanofluids: Theory 33A - Chair: Reza Sadr, Texas A&M University at Qatar

8:00AM G30.00001 Spatial Diffusion of Water in Carbon Nanotubes[1] , AMIR BARATI FARIMANI, N.R. ALURU, Department of Mechanical Science and Engineering, Beckman Institute for Advanced Science and Technology, University of Illinois at Urbana — Water desalination and transport are the great applications of carbon nanotubes (CNTs). Understanding the dynamics of water molecules in carbon nanotubes can shed light on the physics of transport, diffusion and other thermodynamic properties of water. Among all these properties, diffusion is of great importance as it affects most of other key properties. In this article, the spatial variation of the axial, radial, and tangential diffusion coefficients in carbon nanotubes (CNTs) of various diameters were computed. Based on the spatial variation of the diffusion coefficient, the diffusion mechanisms in different regions of the nanotube are defined. The effect of confinement and CNT wall on the diffusion coefficient is studied and discussed. The dependence of the diffusion coefficient on the carbon water interaction parameter is investigated. The average diffusion coefficient in the nanotube as a function of the nanotube diameter is calculated, and the diffusion mechanisms, including the transition regimes, are identified. It has been shown that the axial diffusion coefficient is enhanced in the adjacent water layer to the wall. The results are explained via hydrogen bond network and water orientations.

[1]This research was supported by NSF under grants 0328162, 0810294, 0852657, and 0915718.

8:13AM G30.00002 Transient response at the microchannel-nanochannel interface: chronopotentiometry, chronoamperometry, and electrochemical impedance , JARROD SCHIFFBAUER, YOAV GREEN, SINWOOK PARK, GILAD YOSSIFON, Technion Israel Institute of Technology — Transient response of the interface between a permselective membrane and electrolyte has been studied both theoretically and experimentally in the context of several well-developed electrochemical measurement paradigms. However, such studies of the microchannel-nanochannel interface are conspicuously lacking. One of the fundamental distinctions between the two types of system is the role of convective transport normal to and through the nanochannel. Here we present several recent experimental and theoretical results concerning the transient response of the microchannel-nanochannel interface to a variety of input signals and discuss the relevance of these results in terms of both fundamentals and applications.

8:26AM G30.00003 Orientation selection of block copolymer lamellar phases under oscillatory shear , CHI-DEUK YOO, JORGE VINALS, School of Physics and Astronomy, University of Minnesota — A hydrodynamic description of complex fluids that are structured at the nanoscale necessitates the introduction of appropriate order parameters reflecting the broken symmetries of the fluid, and their coupling to velocity fields and transport. We describe the equations of motion for a uniaxial fluid, and use them to study rheology and orientation selection of a block copolymer under shear. Viscoelastic response is also introduced at this scale, which is shown to lead to local, effective viscoelastic contrast that depends on the local orientation of the lamellae. We further explore domain boundary instabilities that arise from viscoleastic contrast, and their relationship to domain orientation selection in large samples under oscillatory shears.

8:39AM G30.00004 An analytical model of heat transfer in sheared flows of dilute nanofluids[1] , OLEKSII RUDENKO, Eindhoven University of Technology, VICTOR L'VOV, ITAMAR PROCACCIA, Weizmann Institute of Science, FEDERICO TOSCHI, Eindhoven University of Technology — We discuss a model for the enhancement of the heat flux by spherical and elongated (spheroidal) nanoparticles in sheared laminar flows of dilute nanofluids in the presence of a constant temperature gradient. Besides the heat flux carried by the nanoparticles, the model accounts for the contribution of their rotation to the heat flux inside and outside the particles. The rotation of the nanoparticles has a twofold effect: it induces a fluid advection around the particle and it strongly influences the statistical distribution of particle orientations. These dynamical effects are responsible for changing the thermal properties of flowing fluids as compared to quiescent fluids. The proposed model demonstrates a potential increase of the heat flux far beyond the Maxwell-Garnett limit for the spherical nanoparticles, while the spheroidal nanoparticles may either enhance or suppress the heat flux comparing to the spherical nanoparticles.

[1]We acknowledge financial support from the EU FP7 project "Enhanced nano-fluid heat exchange" (HENIX), Contract No. 228882

8:52AM G30.00005 Continuum-based coarse-grained water potentials for structural prediction in confined environments[1] , S.Y. MASHAYAK, N.R. ALURU, Department of Mechanical Science and Engineering, Beckman Institute for Advanced Science and Technology, University of Illinois at Urbana-Champaign — We develop the single site coarse-grained (CG) potential for structural prediction of the confined water. CG potentials allow computationally efficient simulations, which can access larger time- and length-scales compared to fully atomistic simulations. In the literature, various CG techniques have been used to develop CG potentials for structural prediction of bulk water. Due to the inherent inhomogeneity of the confined water system and water's ability to form directional hydrogen-bonds, development of CG potentials for confined water is a formidable task, and not much progress has been done to solve this problem. Herein, we systematically derive the single site CG potentials for the confined water, and show that these potentials can be used in the multi-scale quasi-continuum framework as well as in the CG molecular dynamics (MD) simulations to predict, in a computationally efficient manner, the atomic-level structure of the water confined in two types channels- a silicon slit channel and a graphite slit channel. Our results show that the center of mass density profiles of the water predicted by the CG model are in good agreement with all-atom MD results across multiple length scales, i. e., from few atomic diameters channel widths to 100s of nm.

[1]This work was supported by the NSF under Grants 0328162 (nano-CEMMS, UIUC), 0852657, and 0915718.

9:05AM G30.00006 On the Effects of Brownian particle Movement on the Overall Fluid Velocity Distribution , WAY LEE CHENG, ANOOP BABY, REZA SADR, Texas A&M University at Qatar, MICRO SCALE THERMOFLUIDS LABORATORY TEAM — The suspension of nano-sized particles in a base liquid, known as nanofluids, are reported to display anomalous, often shown enhancement, thermal properties. This enhancement suggests a potential for industrial applications, in particular, for cooling systems. However, the underlying physics that leads to such enhancements in thermal performance is not fully understood. While nano particles/fluid interaction seems to be the main source of the observed phenomena, proposed theories in this regard are often disputed, and not conclusive, pointing to the need for more research in this field. In this study a simple approach is used to study the flow field in nanofluids due to randomly moving Brownian particles in a stationary fluid using numerical simulations. The unconfined Brownian motion of the particles is implemented via Langevin equation. The induced velocity field in the surrounding fluid is obtained by solving the governing hydrodynamic equations accounting for the motion of the particles and statistics of the flow field is then obtained. Effect of other parameters such as temperature and particle density is also investigated.

9:18AM G30.00007 Electron Beam Artifacts in Liquid-Cell Electron Microscopy[1] , JOSEPH GROGAN, University of Pennsylvania, FRANCES ROSS, IBM T. J. Watson Research Center, HAIM BAU, University of Pennsylvania — Liquid-cell electron microscopy has recently emerged as a powerful technique for *in-situ* studies of nanoscale processes in liquids such as nanoparticles' formation, agglomeration, and oriented-assembly and electrochemical plating and imaging of the structure of macromolecules in their native environment. However, many of these phenomena are strongly influenced by the electron beam used for imaging, resulting in artifacts. To utilize the full potential of liquid cell electron microscopy, it is necessary to obtain a good understanding of the interactions of the electron beam with the imaged medium. This study explores the interactions of electrons with water, finding that radiolysis plays a key role in many studies while heating is typically insignificant. We derived a reaction-diffusion model to predict the concentration distribution of radiolysis products H, H_2, OH, H_2O_2, and hydrated electrons, and imaged homogeneous and heterogeneous nucleation of gas bubbles in the liquid cell. The presence of radiolysis products explains many of the observations of crystal formation, growth, and dissolution recently reported in the literature and observed in our laboratory. The study also suggests how to control electron beam effects to suppress or exploit them as desired.

[1]The work was supported, in part, by the NSF grants 1066573 and 1129722.

9:31AM G30.00008 Surface tension relaxation time in liquid-gas and liquid-solid interfaces of simple LJ liquids[1] , ALEX LUKYANOV, ALEXEI LIKHTMAN, University of Reading — We use molecular dynamics (MD) to answer a classical question: how does the surface tension on an interface appear? After defining surface tension from the first principles and conducting several consistency checks, we perform a series of dynamic MD experiments. First, we use a single simple liquid nanodroplet to study liquid-gas (LG) interface dynamics. At time zero, we remove the outer layer of molecules in the droplet, creating a fresh bare interface with the bulk arrangement of molecules. After that the system evolves towards equilibrium, and the expected surface tension is re-established. We found that in the case of LG interfaces, the system relaxation consists of three distinct stages. We have observed this scenario for monatomic Lennard-Jones (LJ) liquids as well as for binary LJ mixtures at different temperatures, monitoring a wide range of physical observables. Second, we use an equilibrated liquid film on a solid substrate. At time zero, we change the strength of the interaction potential between the substrate and liquid molecules and observe how the liquid-solid interface evolves towards equilibrium. We apply results to representative nanoflows over chemically structured substrates and discuss implications to macroscopic modelling of dynamic contact angle.

[1]This work is supported by the EPSRC grant EP/H009558/1.

9:44AM G30.00009 Mobility of a Semiflexible Chain in a Nanochannel , DOUGLAS TREE, University of Minnesota, YANWEI WANG, Soochow University, KEVIN DORFMAN, University of Minnesota — The fundamental understanding of the dynamics of biopolymers in nanoscale devices is an important problem with real-world applicability in high-tech genomic mapping devices. Accordingly, we have developed a comprehensive picture of tube-confined polymers that goes beyond the limiting cases given by the decades-old scaling laws of de Gennes and Odijk for polymer mobility in weak and strong confinement. By using a numerical solution of the confined Green's function and by sampling polymer configurations using a Metropolis Monte Carlo algorithm, we are able to estimate the Kirkwood mobility of long, semiflexible polymers (e.g. DNA) in a square nanochannel with full hydrodynamic interactions. We will present results using this approach that show a broad plateau exists in the mobility as a function of the chain extension for moderate confinement, and that the width of the plateau depends on the anisotropy of the monomers (and hence the ionic strength of the buffer). For dilute, high-ionic strength solutions of DNA, our analysis indicates that the classic results of Odijk and de Gennes apply over a distinctly narrow range of extensions, and we predict that the Rouse-like behavior of moderate confinement will be observed for most of the measurable chain extensions seen in experiments.

Monday, November 19, 2012 8:00AM - 10:10AM –

Session G31 Wind Energy I 33B - Chair: Luciano Castillo, Texas Tech University

8:00AM G31.00001 Grid Sensitivity Analysis of Simulations of a Flow around a Single Rotating Wind Turbine Blade[1] , BRYAN E. KAISER, MICHAEL A. SNIDER, SVETLANA V. POROSEVA, University of New Mexico, ROB O. HOVSAPIAN, Idaho National Laboratory — Design of a wind farm layout with the purpose of optimizing the power outcome requires accurate and reliable simulations of a flow around and behind wind turbines. Such computations are expensive even for a single turbine. To find an optimal set of simulation parameters that satisfies both requirements in simulation accuracy and cost in an acceptable degree, a sensitivity study on how the parameters' variation influences results of simulations should be conducted at the early stage of computations. In the current study, the impact of a grid refinement, grid stretching, and cell shape on simulation results is analyzed in a flow around a single rotating blade utilized in a mid-sized Rim Driven Wind Turbine design (U.S. Patent #7399162) developed by Keuka Energy LLC, and in its near wake. Simulation results obtained with structured and unstructured grids are compared. Industry relies on commercial software for conducting fluid flow simulations. Therefore, STAR-CCM+ software was used in our study. A choice of a turbulence model was made based on our previous sensitivity study of flow simulations over a rotating disk (see M. A. Snider, S. V. Poroseva, AIAA-2012-3146).

[1]Center for Advanced Power Systems, Florida State University

8:13AM G31.00002 Coupling meso- and micro-scale fluid dynamics codes for wind-energy computing[1] , IGNAS SATKAUSKAS, MICHAEL SPRAGUE, MATT CHURCHFIELD, National Renewable Energy Laboratory — Enabled by peta-scale supercomputing, the next generation of computer models for wind energy will simulate a vast range of scales and physics, spanning from wind-turbine structural dynamics and blade-scale turbulence to meso-scale atmospheric flow. This work focuses on new mathematical interface conditions and computational algorithms for coupling meso-scale numerical-weather-prediction codes with micro-scale turbine-vicinity fluid-dynamics codes. Here, an inherent challenge exists when the weather code is based on the compressible Euler equations while the turbine-vicinity flow is modeled by the incompressible Navier-Stokes equations. We propose several one- and two-way code-interaction approaches. These approaches are implemented in a two-dimensional testing platform composed of two in-house codes: (1) a finite-difference code that mimics the weather research and forecasting (WRF) solver and (2) an embedded-domain code based on a common finite-volume approach.

[1]Supported by the Center for Research and Education in Wind

8:26AM G31.00003 LES of turbulent flow past axial flow turbines and turbine arrays: Model development and validation[1] , FOTIS SOTIROPOULOS, SEOKKOO KANG, XIAOLEI YANG, LEONARDO CHAMORRO, CRAIG HILL, University of Minnesota — We present recent progress towards the numerical simulation of turbulent flows past axial-flow wind and hydrokinetic turbines and farms. For simulating multi-turbine arrays, we combine turbine parameterization approaches (actuator disk and actuator line models) with our curvilinear-immersed boundary (CURVIB) LES model. Simulations are carried out both for aligned and staggered wind farms and the computed results are compared with wind tunnel experiments carried out at the St. Anthony Falls Laboratory (SAFL) atmospheric boundary layer wind tunnel. Turbine geometry resolving simulations also employ the CURVIB-LES solver with a wall model and very fine computational grids. Simulations are reported for a complete model marine turbine mounted at the bottom of a straight open channel and the computed results are compared with laboratory experiments obtained in the SAFL Main Channel. The simulated flowfields are analyzed to elucidate the structure of the turbine wake, identify large-scale instabilities, and quantify the mechanisms of turbulence production in the near and far wakes.

[1]This work was supported by US Department of Energy (Grant No. DE-EE0002980, DE-EE0005482), Xcel Energy (Grant No. RD3-42), Verdant Power, Initiative for Renewable Energy & the Environment (Grant No. RO-0004-12), and Minnesota Supercomputing Institute.

8:39AM G31.00004 The Penn State "Cyber Wind Facility" , JAMES BRASSEUR, GANESH VIJAYAKUMAR, ADAM LAVELY, TARAK NANDI, BALAJI JAYARAMAN, PANKAJ JHA, ALEX DUNBAR, JAVIER MOTTA-MENA, Penn State, SUE HAUPT, NCAR, BRENT CRAVEN, ROBERT CAMPBELL, SVEN SCHMITZ, ERIC PATERSON, Penn State — We describe development and results from a first generation Penn State "Cyber Wind Facility" (CWF). The aim of the CWF program is to develop and validate a computational "facility" that, in the most powerful HPC environments, will be basis for the design and implementation of cyber "experiments" at a level of complexity, fidelity and resolution to be treated similarly to field experiments on wind turbines operating in true atmospheric environments. We see cyber experiments as complimentary to field experiments in the sense that, whereas field data can record over ranges of events not representable in the cyber environment, with sufficient resolution, numerical accuracy, and HPC power, it is theoretically possible to collect cyber data from more true, albeit canonical, atmospheric environments can produce data from extraordinary numbers of sensors impossible to obtain in the field. I will describe our first generation CWF, from which we have quantified and analyzed useful details of the interactions between atmospheric turbulence and wind turbine loadings for an infinitely stiff commercial-scale turbine rotor in a canonical convective daytime atmospheric boundary layer over horizontally homogeneous rough flat terrain. Supported by the DOE Offshore Initiative and the National Science Foundation.

8:52AM G31.00005 Proper Orthogonal Decomposition analysis of kinetic energy entrainment in large wind farms[1] , CLAIRE VERHULST, CHARLES MENEVEAU, Johns Hopkins University — Vertical entrainment of kinetic energy is thought to play an important role in the dynamics of very large wind farms (Calaf et al., Phys Fluids 2010; and Cal et al. J. Ren. Sust. Energy 2010). To elucidate dominant mechanisms and flow physics of this vertical transfer of kinetic energy, we use Proper Orthogonal Decomposition (POD) to extract dominant flow structures from snapshots of velocity fields generated using Large Eddy Simulation of flow in an infinite turbine array in the atmospheric boundary layer. The POD analysis shows that the dominant modes are large streamwise counter-rotating vortices located above the turbines. The contribution of each POD mode to kinetic energy entrainment at the turbine level is then quantified and the modes are ordered by this contribution. Interestingly, the number of POD modes needed to represent dominant portions of the kinetic energy entrainment is less that the number needed to represent similar portions of the kinetic energy in the turbulent field. This suggests that understanding and controlling only a small number of flow structures may be relevant to the design of very large wind farms. In addition, to understand how the array layout affects the POD modes, several turbine orientations (aligned, staggered, etc) will be discussed.

[1]This research is supported by a NSF Graduate Fellowship and by NSF-CBET 1133800. Computer time by NCAR is also appreciated.

9:05AM G31.00006 An improved effective roughness height model for optimization of wind farm layout[1] , XIAOLEI YANG, FOTIS SOTIROPOULOS, St. Anthony Falls Laboratory, Department of Civil Engineering, University of Minnesota, 2 Third Avenue SE, Minneapolis, MN 55414, USA — An improved effective roughness height model is developed to account for the different effects of streamwise turbine spacing and spanwise turbine spacing, which cannot be well captured by the classic model when the ratio of streamwise spacing to spanwise spacing is not equal to 1. The central idea of the present model is approximating the nominal incoming velocity by a kinematic model, which is the time- and horizontal-averaged velocity at hub height in the classic model. The prediction capability of the present model is validated by comparing with the results from large-eddy simulation of infinite large aligned wind farms. The present model is also tested for finite-size wind farms and staggered wind farms.

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

9:18AM G31.00007 Wind Turbine Wakes with Actuator Line Aerodynamics , YULIA PEET, Arizona State University — Actuator line aerodynamics (AL) model is becoming increasingly popular for characterization of the flow field and the turbulent wake created by the rotated turbines. AL model does not require boundary layer resolution and is thus significantly more efficient than the fully-resolved computations. Thus, simulation of multiple wind turbines and characterization of turbulent wakes in such multiple-turbine configurations is possible with the current model. In this talk, we investigate the properties of wind turbine wakes calculated by Large Eddy Simulations with the actuator line model, as a function of several parameters, including Reynolds number, tip speed ratio and distance between the turbines. Spectral element fluid dynamics code Nek5000 is used for the simulations.

9:31AM G31.00008 Computational investigation of hydrokinetic turbine arrays in an open channel using an actuator disk-LES model[1] , SEOKKOO KANG, XIAOLEI YANG, FOTIS SOTIROPOULOS, University of Minnesota — While a considerable amount of work has focused on studying the effects and performance of wind farms, very little is known about the performance of hydrokinetic turbine arrays in open channels. Unlike large wind farms, where the vertical fluxes of momentum and energy from the atmospheric boundary layer comprise the main transport mechanisms, the presence of free surface in hydrokinetic turbine arrays inhibits vertical transport. To explore this fundamental difference between wind and hydrokinetic turbine arrays, we carry out LES with the actuator disk model to systematically investigate various layouts of hydrokinetic turbine arrays mounted on the bed of a straight open channel with fully-developed turbulent flow fed at the channel inlet. Mean flow quantities and turbulence statistics within and downstream of the arrays will be analyzed and the effect of the turbine arrays as means for increasing the effective roughness of the channel bed will be extensively discussed.

[1]This work was supported by Initiative for Renewable Energy & the Environment (IREE) (Grant No. RO-0004-12), and computational resources were provided by Minnesota Supercomputing Institute.

9:44AM G31.00009 Vertical Mean Kinetic Energy Entrainment in a Scaled Wind Turbine Array , ANDREW NEWMAN, Texas Tech University, DON DREW, Rennselaer Polytechnic Institute, LUCIANO CASTILLO, Texas Tech University — A 2D model of the Mean Kinetic Energy (MKE) of a scaled wind turbine array was analyzed to understand how turbulent transport brings MKE into arrays from the Turbulent Boundary Layer above. This was done by applying a Proper Orthogonal Decomposition to particle image velocimetry data and constructing modal expansions for the Reynolds stress terms which appear in the transport equation along a horizontal surface above the array. These terms have been shown to be of the same order of magnitude as the power extracted from the turbines. It was also found that 75% of the total Reynolds shear stress was carried by the first 13 modes. A strong correlation between a mode's Reynolds shear stress and its contribution to the MKE entrainment was demonstrated. Thus, a small number of modes are responsible for a large quantity of the MKE entrainment for the array. Modal streamwise length scales were determined; it was found that modal length decreases with increasing mode number. By considering sums of modes the largest scales observable in the experiment (13 rotor diameters) were shown to contribute over 50% of the MKE entrainment.

9:57AM G31.00010 Measurements in an axisymmetric turbulent wake with rotation downstream of a model wind turbine , NATHANIEL DUFRESNE, MARTIN WOSNIK, University of New Hampshire — Energy production data from several of the existing offshore wind farms indicate that turbine arrays may enter a stall condition which can cause an overall energy production shortfall(which can exceed 10%). This deep array stall is (presumably) due to the wakes generated by turbines upstream interacting with turbine rotors downstream. It is hypothesized that there is a critical array spacing at which this stall occurs, but that this spacing is dependent on rotor thrust c_T (which is determined by tip-speed ratio λ and power coefficient c_P of the rotor), Reynolds number, upstream conditions, and possibly wall roughness. An experimental investigation of the axial and azimuthal velocity field measurements in the wake of a single 3-bladed wind turbine with rotor diameter of 0.91m was conducted. The turbine was positioned in the free stream, near the entrance of the 6m x 2.5m test section of the UNH FPF, which can achieve test section velocities of up to 15 m/s and Reynolds numbers $\delta^+ = \delta u_\tau/\nu \approx 30,000$. Hot-wire anemometry was used to obtain velocity field measurements. The data obtained will be used to examine similarity scaling functions for velocity, wake growth, and turbulence derived from an equilibrium similarity analysis of the far wake.

Monday, November 19, 2012 8:00AM - 10:10AM –

Session G32 Granular Flows I 33C - Chair: Yi Fan, Northwestern University

8:00AM G32.00001 A modified kinetic theory for frictional granular flows in dense and dilute regimes , SEBASTIAN CHIALVO, SANKARAN SUNDARESAN, Department of Chemical and Biological Enginering, Princeton University — We investigate the rheology of granular materials in both the dense and dilute inertial regimes via molecular dynamics simulations of homogeneous, simple shear flows of soft, frictional spheres. Though traditional kinetic theories are often used for continuum modeling of such materials, they fail to describe flow behavior in dense systems near the jamming transition and do not account explicitly for interparticle friction. On the basis of our simulations, we propose a new model for the radial distribution function at contact as well as modifications to the shear stress and energy dissipation equations of one commonly used theory [1]. These changes account for stress and temperature scalings observed in our steady shear simulations while preserving the dynamic nature of the kinetic theory model.

[1] V. Garzó and J.W. Dufty, Phys. Rev. E 59, 5895 (1999).

8:13AM G32.00002 Homogeneous Cooling Granular Gases of Cohesive Particles[1] , ERIC MURPHY, SHANKAR SUBRAMANIAM, Iowa State University — We consider the case of a homogeneously cooling gas of dissipative granular particles with the addition of short-range attractive potentials. An analytic solution is found using the pseudo-Liouville formalism in terms of a nondimensional ratio of interparticle potential energy to internal energy of the system. The solution reveals that the granular temperature evolution is indistinguishable from Haff's law until a critical temperature region is approached. In this critical region, an abrupt increase in cooling and aggregation are predicted. Lastly, the solution is compared against soft-sphere DEM data. The abrupt increase in cooling sheds light on the expected rheological behavior and jamming transitions in flows of such particles.

[1]NSF Award Number 0927660

8:26AM G32.00003 Axisymmetric Column Collapse in a Rotating System , JAY WARNETT, PETER THOMAS, PETR DENNISENKO, University of Warwick UK — We discuss experimental and computational results of a study investigating the collapse of an initially axisymmetric cylindrical column of granular material within a rotating environment of air or liquids. In industry this type of granular column collapse that is subject to background rotation is encountered, for instance, in the context of the spreading of powders and fertilizers. In comparison to its non-rotating counterpart the physical characteristics of the column collapse in a rotating system are expected to be modified by effects arising from centrifugal forces and Coriolis forces. We compare our new results for the rotating flow to data available in the literature for the collapse of granular columns in non-rotating systems to highlight the differences observed.

8:39AM G32.00004 On the Collapse of Granular Columns in Different Gravities , HORACIO TAPIA-MCCLUNG, ROBERTO ZENIT, National Autónomous University of México — By performing numerical simulations of the collapses of granular columns we find that the scaling of the final height of the emplacements ($\sim a^\beta$) is preserved when the inter-granular friction coefficient and the initial aspect ratio of the columns is varied under different gravitational accelerations. The top of the column initially evolves closely to the free falling law for large aspect ratios and gravities. For high aspect ratio columns in low gravities, an initial fluidization of the grains is observed. We present energy balances during the emplacement and measurements of a quantity equivalent to the inertial number, to understand the influence of varying the gravitational acceleration on the properties of the column collapses.

8:52AM G32.00005 Collapse of a granular column: discrete element simulations and continuum modelling , LAURENT LACAZE, EDOUARD IZARD, Institut de Mécanique des Fluides de Toulouse, RICH KERSWELL, University of Bristol — The unsteady dynamics of the collapse of a granular column onto a horizontal plane exhibits a wealth of interesting behaviour typical of granular flows. This canonical flow situation has therefore received attention for the last decade through different experimental and numerical studies. Using 3-dimensional soft particles simulations, the observed behaviour can be faithfully reproduced and the observed scaling laws for the final deposit captured. A coarse-graining procedure is then used to extract models for both the apparent rheology and slip velocity at the base. These are incorporated into a "Saint-Venant"-type code to model unsteady granular collapse in the limit of small aspect ratio.

9:05AM G32.00006 The hour-glass: comparisons of discrete granular flow and continuum plastic flow. , PIERRE-YVES LAGREE, LYDIE STARON, CNRS, AURELIEN GRABSH, ENS Cachan, STEPHANE POPINET, NIWA, D'ALEMBERT COLLABORATION, NIWA COLLABORATION — A hour-glass is a fascinating way to measure time, surprisingly the flow is not function of the filling height as in a clepsydra. The discharge of a granular silo implies a constant rate, dictated by the size of the aperture, but independent of the height of material stored (the Berveloo law). This observation is often understood as resulting from the friction forces mobilized at the walls of the silo, which decrease the apparent weight of the material, and screen the bottom area from the pressure, (Janssen effect). This explanation fails however in the case of wide systems for which walls are distant from several times the height of material stored. In this contribution, we simulate the continuum counterpart of the granular silo by implementing the plastic (I)-rheology in a 2D Navier-Stokes solver (Gerris) and compare with Contact Dynamics simulations. Velocity field and the pressure field are compared and discussed in the light of the two simulation methods. We recover the Berverloo scaling relating discharge rate and aperture size. This result points at the existence of a yield stress, rather than at the mobilization of friction forces at walls, as controlling the discharge of the granular silo.

9:18AM G32.00007 A study of Force chain statistics in quasi-2D granular systems , JIE ZHANG, Institute of Natural Sciences and Department of Physics, Shanghai Jiao Tong University, Shanghai 200240, China, LING ZHANG, YUJIE WANG, Department of Physics, Shanghai Jiao Tong University, Shanghai 200240, China, ROBERT ECKE, Center for Nonlinear Studies, Los Alamos National Laboratory, Los Alamos, NM 87545, USA, ROBERT BEHRINGER, Department of Physics, Duke University, Durham, NC 27708, USA — Force chains play a key role in the understanding of mechanical properties of granular materials. In this study, we have examined several statistical properties of force chains in a number of different granular systems using bi-disperse photo-elastic disks. These systems include vertical slabs of granular materials under the gravitational field, horizontal layers of granular materials under isotropic compression and under pure shear. Despite of drastically different protocols and processes used to generate these systems, we have found that there is a universal distribution of force chain length: they all obey exponential –like distributions. The exponential distributions can be explained using a diffusion-like argument. Despite the success of this argument, a fundamental question remains: how can force chains, which are "believed" to be *hyperbolic* in nature, have their lengths obey exponential distributions, which are intrinsically diffusion-like?

9:31AM G32.00008 Kinematics of segregating granular mixtures in quasi-2D heaps[1] , YI FAN, PAUL UMBANHOWAR, JULIO OTTINO, RICHARD LUEPTOW, Northwestern University, Evanston, IL — Segregation of granular mixtures of different sized particles in heap flow appears in a variety of contexts. Our recent experiments showed that when bi-disperse mixtures of different sized spherical particles fill a quasi-two dimensional (2D) silo, three different final heap configurations – stratified, segregated, and mixed – occur, depending on either 2D flow rate or heap rise velocity. However, since it is difficult to measure the kinematic details of the segregating granular mixtures in heap flow experimentally, the underlying mechanisms for how 2D flow rate or heap rise velocity influences final particle configurations have not been well understood. In this work, we use the discrete element method (DEM) to simulate heap flow of bi-disperse mixtures in experimental scale quasi-2D heaps. The final particle distributions in the simulations agree quantitatively with experiments. We measure several key kinematic properties of the segregating granular mixtures including the local flow rate, velocity, and flowing layer thickness. We correlate the characteristics of these kinematic properties with the local particle distributions of the mixtures. This provides new insights for understanding the mechanisms of segregation and stratification in heap flow including the linear decrease in flow rate and maximum velocity down the heap as well as the relatively constant flowing layer thickness along the length of the heap.

[1]Funded by Dow Chemical Co.

9:44AM G32.00009 Stratification, segregation and mixing of bi-disperse granular materials in quasi-2D heaps[1] , RICHARD LUEPTOW, YI FAN, PAUL UMBANHOWAR, JULIO OTTINO, Northwestern University, Evanston, IL — Segregation and mixing of granular mixtures during heap formation have important consequences across a range of contexts, from chemical processing to construction to agriculture. This research investigates three different final particle configurations of bi-disperse granular mixtures of spherical particles - stratified, segregated and mixed - during filling of quasi-two dimensional silos. We considered a larger number and relatively wider range of control parameters than previous studies, including particle size ratio, flow rate, system size, and heap rise velocity. The boundary between the stratified and unstratified states is primarily controlled by the two-dimensional flow rate, with the critical flow rate for the transition depending weakly on particle size ratio and flowing layer length. In contrast, the transition from segregated to mixed states is controlled by the rise velocity of the heap, a control parameter not previously considered. The critical rise velocity for the transition from a segregated state to a mixed state depends strongly on the particle size ratio.

[1]Funded by Dow Chemical Co.

9:57AM G32.00010 Resolving a paradox of anomalous scalings in the diffusion of granular materials[1] , IVAN C. CHRISTOV, HOWARD A. STONE, Department of Mechanical and Aerospace Engineering, Princeton University — Granular materials do not perform Brownian motion, yet diffusion can be observed in such systems when agitation causes inelastic collisions between particles. It has been suggested that axial diffusion of granular matter in a rotating drum might be "anomalous" in the sense that the mean squared displacement of particles follows a power law in time with exponent less than unity. Further numerical and experimental studies have been unable to definitively confirm or disprove this observation. We show two possible resolutions to this apparent paradox without the need to appeal to anomalous diffusion. First, we consider the evolution of arbitrary (non-point-source) initial data towards the self-similar intermediate asymptotics of diffusion by deriving an analytical expression for the instantaneous collapse exponent of the macroscopic concentration profiles. Second, we account for the concentration-dependent diffusivity in bidisperse mixtures, and we give an asymptotic argument for the self-similar behavior of such a diffusion process, for which an exact self-similar analytical solution does not exist. The theoretical arguments are verified through numerical solutions of the governing partial differential equations.

[1]This work was supported, in part, by NSF Grant DMS-1104047.

Monday, November 19, 2012 8:00AM - 10:10AM –

Session G33 Mini-Symposium: Complex Fluid Flows in Memory of Daniel D. Joseph I 29A - Chair: Howard Hu, University of Pennsylvania

8:00AM G33.00001 Daniel D. Joseph, a Life in Fluid Dynamics , JAMES FENG, University of British Columbia — I will give an overview of the remarkable career of Dan Joseph, and highlight the greatest contributions he has made to our discipline.

8:13AM G33.00002 Loss of deformability of malaria-infected red blood cells , S. MAJID HOSSEINI, JAMES FENG, University of British Columbia — The pathogenesis of malaria is largely due to stiffening of the infected red blood cells (RBCs). Contemporary understanding ascribes the loss of RBC deformability to a 10-fold increase in membrane stiffness caused by extra cross-linking in the spectrin network. Local measurements by micropipette aspiration, however, have reported only an increase of 3-fold in the shear modulus. We believe the discrepancy stems from the rigid parasite particles inside infected cells, and have carried out numerical simulations to demonstrate this mechanism. The cell membrane is represented by a set of discrete particles connected by linearly elastic springs. The cytosol is modeled as a homogeneous Newtonian fluid, and discretized by particles as in standard smoothed particle hydrodynamics. The malaria parasite is modeled as an aggregate of particles constrained to rigid-body motion. We simulate RBC stretching tests by optical tweezers in three dimensions. The results demonstrate that the presence of a sizeable parasite greatly reduces the ability of RBCs to deform under stretching. With the solid inclusion, the observed loss of deformability can be predicted quantitatively using the local membrane elasticity measured by micropipettes.

8:26AM G33.00003 Thermal convection, energy stability theory, viscous fingering, and capillary attraction: a 40 year interaction with Dan Joseph , G.M. HOMSY, Depts. of Mathematics and of Mechanical Engr., UBC — Dan Joseph made major contributions to all the topics in the title. These topics are also areas of interaction between Dan and the speaker that began in 1969 and continued for 40+ years. This talk will report on our recent work on capillary attraction between floating particles and, as time allows, will include some reminiscences.

8:52AM G33.00004 Some surprising results on convective transport in the Sun , KATEPALLI SREENIVASAN, New York University, SHRAVAN HANASOGE, Princeton University, THOMAS DUVALL, JR., NASA — Convection in the interior of our Sun comprises structures on a wide spectrum of scales. The dynamical parameters relevant to the problem cannot be replicated in laboratory experiments or numerical simulations, so our understanding of these observed features is quite incomplete even at the phenomenological level. We analyze observations of the wavefield in solar photosphere using techniques of time-distance helioseismology to deduce flows in the solar interior. We downsample and synthesize 900 billion wavefield observations to produce 3 billion cross-correlations, which we average and fit, measuring 5 million wave travel times. Using these travel times, we deduce the underlying flow systems and study their statistics to bound convective velocity magnitudes in the solar interior, as a function of depth and spherical-harmonic degree. We find the convection velocities so deduced to be 20-100 times weaker than suggested by current theoretical estimates. This result indicates the prevalence of a different paradigm of turbulence from that predicted by existing models, prompting the question: what mechanism transports the heat flux of a solar luminosity outwards? We cast our results in terms that should be particularly relevant to turbulent transport in convective systems with rotation.

9:18AM G33.00005 Forming adjustable monolayers via particle assembly at electrified liquid-fluid interfaces , NADINE AUBRY, Carnegie Mellon University, PUSHPENDRA SINGH, New Jersey Institute of Technology, MUHAMMAD JANJUA, SAI NUDURUPATI, Lake Superior State University — The application of an external electric field leads to the assembly of particles at a liquid-fluid interface into monolayers which display long-range order and whose spacing between the particles can be adjusted by varying the strength and/or frequency of the electric field. In contrast to capillarity induced self-assembly, the technique permits the assembly of *small* particles, i.e., submicron to nano sized particles. This is possible because (i) the particles experience electric field induced capillary forces and (ii) the associated energy of such forces is greater than kT. The adjustment of the lattice spacing, and therefore the control of the mechanical, thermal, optical properties of the monolayer, is achieved through a judicious combination of attractive capillary forces and repulsive particle-particle interactions which is realized in practice by varying the electric field.

9:44AM G33.00006 Dan Joseph's contributions to disperse multiphase flow[1] , ANDREA PROSPERETTI, Mechanical Engineering, Johns Hopkins Universitry and Applied Sciences, University of Twente — During his distinguished career, Dan Joseph worked on a vast array of problems. One of these, which occupied him off and on over the last two decades of his life, was that of flows with suspended finite-size particles at finite Reynolds numbers. He realized early on that progress in this field had to rely on the insight gained from numerical simulation, an area in which he was a pioneer. On the basis of the early numerical results he recognized the now famous "drafting, kissing and tumbling" mechanism of particle-particle interaction, the possibility of fluidization by lift and many others. With a number of colleagues and a series of gifted students he produced a significant body of work summarized in his on-line book *Interrogations of Direct Numerical Simulation of Solid-Liquid Flows* available from `http://www.efluids.com/efluids/books/joseph.htm`. This presentation will describe Joseph's contribution to the understanding of disperse multiphase flow and conclude with some examples from the author's recent work in this area.

[1]Supported by NSF.

Monday, November 19, 2012 10:30AM - 12:27PM –

Session H1 Geophysical: Ocean II 22 - Chair: Andres Tejada-Martinez, University of South Florida

10:30AM H1.00001 Large-eddy simulation of open channel flow with surface cooling , RACHEL WALKER, ANDRES TEJADA-MARTINEZ, University of South Florida, GUILLAUME MARTINAT, CHESTER GROSCH, Old Dominion University — Results are presented from large-eddy simulations of unstably stratified open channel flow driven by a pressure gradient with zero surface shear stress and a no-slip bottom. Unstable stratification is imposed by a constant cooling flux at the surface and an adiabatic bottom wall. Under neutrally stratified conditions, the flow is characterized by weak full-depth streamwise cells similar to, but less coherent than Couette cells in plane Couette flow. Surface cooling leads to full-depth convection cells characterized by greater coherency than Couette cells. The structure of the turbulence and turbulence statistics are analyzed with respect to the Rayleigh number (Ra) representative of the surface buoyancy relative to shear. Increased surface cooling and thus increased Ra leads to full-depth convection cells of greater spanwise size than Couette cells. Impact of the convection cells on mean velocity, root mean square of velocity, and budgets of resolved turbulent kinetic energy and Reynolds shear stress will be investigated as a function of Rayleigh number. These results motivate further studies of the effect of surface cooling on tidal boundary layers simulated via an oscillating pressure gradient.

10:43AM H1.00002 Evaluation of turbulence models in RANSS of wind-driven flow with full-depth Langmuir circulation , NITYANAND SINHA, ANDRES E. TEJADA-MARTINEZ, University of South Florida, CHESTER E. GROSCH, GUILLAUME MARTINAT, Old Dominion University — Large-eddy simulations of full-depth Langmuir circulation (LC) in a wind-driven shear current in neutrally-stratified shallow water have revealed modified bottom log-layer dynamics. For example, mixing due to LC induces a large wake region eroding the classical log-law velocity profile within the range $90 < z+ < 200$. This has important implications on Reynolds-averaged Navier-Stokes simulations (RANSS) of the coastal ocean circulation. Turbulence models in RANSS are typically calibrated under the assumption of log-layer dynamics, which could potentially be invalid during occurrence of full-depth LC, often observed in shallow coastal shelf regions. Motivated by this, we perform RANSS of wind-driven flows with LC with a 1-D water column model in order to assess the performance of various turbulence parameterizations such as the k-epsilon model in capturing the disruption of log-layer dynamics induced by LC. Modifications to these models will be described in order to account for the presence of LC.

10:56AM H1.00003 Scalar transport in large-eddy simulation of Langmuir turbulence in shallow water , ANDRES E. TEJADA-MARTINEZ, University of South Florida, CIGDEM AKAN, Oregon State University, CHESTER GROSCH, GUILLAUME MARTINAT, Old Dominion University — Large-eddy simulations (LES) of wind-driven shallow water flows with Langmuir turbulence have been conducted and associated passive scalar transport analyzed. In these flows, the largest scales of the Langmuir turbulence consist of full-depth Langmuir circulation (LC), parallel downwind-elongated, counter-rotating vortices acting as a secondary structure to the mean flow. Langmuir turbulence is generated by the interaction of the wind-driven shear current with the Stokes drift velocity induced by surface gravity waves. LES shows that Langmuir turbulence plays a major role in determining scalar transport throughout the entire water column and scalar transfer at the surface. Langmuir turbulence affects scalar transport and its surface transfer through 1. the full-depth homogenizing action of the large-scale LC and 2. the near-surface vertical turbulence intensity induced by the Stokes drift velocity shear. Results from simulations are analyzed in order to understand the effect of wind and wave forcing parameters on these two mechanisms.

11:09AM H1.00004 Numerical Simulations of an Asymptotically Reduced Model of Anisotropic Langmuir Turbulence , ZHEXUAN ZHANG, GREGORY CHINI, University of New Hampshire, KEITH JULIEN, University of Colorado, Boulder, EDGAR KNOBLOCH, University of California, Berkeley — The Craik–Leibovich (CL) model of Langmuir circulation (LC), a prominent form of shear turbulence in the ocean surface boundary layer (BL), is a variant of the Navier–Stokes equations in which high-frequency surface wave dynamics are filtered. Various investigators have performed large-eddy simulations of the CL equations in moderate-sized domains. Extension of these simulations to wide domains, several hundred times the BL depth, is computationally intensive, yet is necessary for investigating the impact of LC on submesoscale upper ocean phenomena. To facilitate such simulations, Chini et al. (GAFD, 2009) derived asymptotically exact, reduced CL equations by exploiting the strongly anisotropic character of LC in the strong CL vortex-force limit using multiscale analysis. The reduced equations go beyond strictly 2D (downwind invariant) formulations by consistently incorporating the dominant 3D physical processes while averaging or filtering certain fast, fine-scale flow features. Here, secondary stability analysis and pseudospectral numerical simulations are performed to explore the dynamics of the reduced model. The simulations suggest a possible dynamical explanation for the commonly observed Y-junctions in LC surface signatures.

11:22AM H1.00005 Numerical Simulations of a Multiscale Model of Stratified Langmuir Circulation[1] , ZIEMOWIT MALECHA, IAM, Univeristy of New Hampshire, GREGORY CHINI, Department of Mechanical Engineering, University of New Hampshire , KEITH JULIEN, Applied Mathematics, University of Colorado Boulder — Langmuir circulation (LC), a prominent form of wind and surface-wave driven shear turbulence in the ocean surface boundary layer (BL), is commonly modeled using the Craik–Leibovich (CL) equations, a phase-averaged variant of the Navier–Stokes (NS) equations. Although surface-wave filtering renders the CL equations more amenable to simulation than are the instantaneous NS equations, simulations in wide domains, hundreds of times the BL depth, currently earn the "grand challenge" designation. To facilitate simulations of LC in such spatially-extended domains, we have derived multiscale CL equations by exploiting the scale separation between submesoscale and BL flows in the upper ocean. The numerical algorithm for simulating this multiscale model resembles super-parameterization schemes used in meteorology, but retains a firm mathematical basis. We have validated our algorithm and here use it to perform multiscale simulations of the interaction between LC and upper ocean density stratification.

[1]ZMM, GPC, KJ gratefully acknowledge funding from NSF CMG Award 0934827

11:35AM H1.00006 POD analysis of Langmuir circulation interacting with a crossed pressure gradient driven flow , GUILLAUME MARTINAT, CHESTER GROSCH, Center for Coastal Physical Oceanography / Old Dominion University, ANDRES TEJADA-MARTINEZ, University of South Florida — The interaction between surface gravity wave and a wind driven shear current can lead to the generation of longitudinal counter-rotating pairs of vortices. This phenomenon is known as Langmuir Circulation and can extend to the full depth of the water column in coastal shallow waters. In cases of full depth Langmuir circulation, the cells are subjected to interactions with tidal currents of comparable magnitude as the current generated by wind shear and the intensity of the cells may be affected. The intent of this study is to apply proper orthogonal decomposition on results obtained through large eddy simulation, in order to evaluate the influence of a cross tidal current on the energy contained in Langmuir cells with a wave forcing varying in amplitude and wavelength. The stress Reynolds numbers considered will allow an equal stress for the pressure gradient driven flow and the wind driven flow and the total Reynolds number is set at 395.

11:48AM H1.00007 Langmuir circulation in sheared shallow waters[1] , WILLIAM PHILLIPS, Swinburne University of Technology — The instability of shallow water waves on a moderate shear to Langmuir circulation is considered. In such instances, specifically at the shallow end of the inner coastal region, the shear can significantly affect the drift giving rise to profiles markedly different from the simple Stokes drift. Since drift and shear are instrumental in the instability to Langmuir circulation, of key interest is how that variation in turn affects onset to Langmuir circulation. Also of interest is the effect to onset of various boundary conditions, viz Neumann and Cauchy. Two typical flow fields are considered, namely shear driven and current driven flow. Relative to the reference case, shear driven flow is found to be destabilizing and current driven stabilizing to Langmuir circulation. In current driven flow it is further found that multiple layers, as opposed to a single layer, of Langmuir circulation can form. Moreover the layers can extend into a region of flow beyond that in which the instability applies. Finally, while Neumann-Neumann are known to ensure the least stable spanwise wavenumber is zero and Cauchy-Neumann boundary conditions non zero, we find the latter further act to realize the long observed but unexplained large aspect-ratio shallow water Langmuir circulation.

[1]NSF OCE-0116921; ARC DP-1093517

12:01PM H1.00008 Interfacial flux in wetlands predicted using surface divergence measurements , CRISTINA POINDEXTER, EVAN A. VARIANO, University of California, Berkeley — Surface divergence has been shown to be a robust predictor of the air-water gas transfer velocity, k. We used this surface divergence model to investigate the effects of wind on k in wetlands with emergent vegetation. We used fluoropolymer tubes to represent plant stems in a laboratory tank equipped with a wind tunnel. The fluoropolymer material provided optical access to the water flows directly around the "stems" for PIV. The k values predicted by the surface divergence model from PIV-derived near surface divergence fields in the tank matched directly-measured k values in the tank. The surface divergence fields also illustrated a mechanism for wind-induced gas transfer in wetlands with emergent vegetation. We observed an area of high surface divergence surrounding each stem and order of magnitude lower surface divergence in areas away from any stems. Thus we expect a nearly linear relationship between stem density and k (if average wind speed in the emergent canopy is held constant). The agreement between modeled and measured k values in this low-Reynolds-number, obstructed flow provides further support for the universality of the surface divergence model for k. The results also permit improved prediction of k in wetlands.

12:14PM H1.00009 On similarity of wind-waves spectral shapes in laboratory and in ocean , LEV SHEMER, ANDREY ZAVADSKY, DAN LIBERZON, Tel-Aviv University — Wind-wave field evolving in a compact laboratory facility that consists of a wind tunnel capable of generating wind speed that may exceed 15 m/s atop of a 5 m long wave tank is studied. Surface elevation measurements were carried out at numerous positions along the test section and at different mean wind flow rates. For each experimental condition, the accumulated records were long enough to contain at least $O(10^4)$ dominant waves; the wave power spectra computed from the recorded time series cover up to 5 decades. Similarity of the spectral shapes in the vicinity of the peak frequency f_p obtained at various fetches and wind conditions was observed. This similarity manifests itself when normalized frequency deviation from the peak value is introduced. Detailed comparison is carried out of spectra obtained in the present measurements at all fetches and wind conditions, to the similarly normalized JONSWAP spectrum that represents field experiments. When estimating spectral tail behavior, care was taken to consider frequencies exceeding about $3.5f_p$ to alleviate the effect of bound waves. The spectral tails dependence on frequency follows the power law f^{-n}; the values of $n$being in the range $3 < n < 4$, depending on wave age. It thus can be concluded that the spectra of wind-waves in a small facility exhibit significant similarities to those obtained in field studies at much larger scales, as well as to theoretical estimates.

Monday, November 19, 2012 10:30AM - 12:27PM –

Session H2 Convection and Buoyancy-Driven Flows V: Exchange Flows 23A - Chair: Ross Griffiths, Australian National University

10:30AM H2.00001 Internal Waves Generated by Mixed Region Collapse in the Ocean , AMBER HOLDSWORTH, BRUCE SUTHERLAND, University of Alberta — Tropical cyclones are known to mix the relatively warm near-surface fluid with the cooler underlying fluid creating a well-mixed region of uniform density. The well-mixed region collapses into the stably stratified ambient and forms an intrusive gravity current. This motion is a mechanism for the generation of downward propagating internal waves. We will present a series of laboratory experiments used to examine the axisymmetric collapse of a well-mixed region in a uniformly stratified and rotating fluid. A square tank was filled with uniformly stratified fluid and a hollow cylinder of radius $R_c \sim 5\,$cm and depth H_m between 5 and 15 cm was suspended at the center of the tank. Synthetic Schlieren was used to determine wave characteristics such as the frequency ω and radial wavenumber k_r. We found that internal wave frequencies were set by the buoyancy frequency $\omega \approx 0.75N$, the radial wavenumber scaled with R_c so that $k_r R_c \approx 3.5$ and the vertical displacement amplitude increases with H_m. To explore a wider range of parameters the experimental data was used to calibrate a numerical model of the axisymmetric collapse. We examined the effects of changing the aspect ratio of the lock R_c/H_m and the Rossby number Ro.

10:43AM H2.00002 On the spreading and instability of gravity current fronts of arbitrary shape[1] , N. ZGHEIB, Department of Mechanical and Aerospace Engineering, University of Florida, Gainesville, Florida, USA., T. BONOMETTI, Université de Toulouse, INPT, UPS, Institut de Mécanique des Fluides de Toulouse, Allée Camille Soula, F-31400 Toulouse, France, S. BALACHANDAR, Department of Mechanical and Aerospace Engineering, University of Florida, Gainesville, Florida, USA — Experiments, simulations and theoretical analysis were carried out to study the influence of geometry on the spreading of gravity currents. The horizontal spreading of three different intial planforms of initial release were investigated: an extended ellipse, a cross, and a circle. The experiments used a pulley system for a swift nearly instantaneous release. The case of the axisymmetric cylinder compared favorably with earlier simulations. We ran experiments for multiple aspect ratios for all three configurations. Perhaps the most intriguing of the three cases is the "ellipse," which within a short period of release flipped the major and minor axes. This behavior cannot be captured by current theoretical methods (such as the Box Model). These cases have also been investigated using shallow water and direct numerical simulations. Also, in this study, we investigate the possibility of a Rayleigh-Taylor (RT) instability of the radially moving, but decelerating front. We present a simple theoretical framework based on the inviscid Shallow Water Equations. The theoretical results are supplemented and compared to highly resolved three-dimensional simulations with the Boussinesq approximation.

[1]Chateaubriand Fellowship – NSF PIRE grant OISE-0968313

10:56AM H2.00003 Filling box stratification fed by a gravity current , CHARLIE HOGG, HERBERT HUPPERT, Institute of Theoretical Geophysics, University of Cambridge, JORG IMBERGER, Centre for Water Research, University of Western Australia — Fluids in confined basins can be stratified by the filling box mechanism. The source of dense fluid in geophysical applications, such as a cold river entering a warmer lake, can be a gravity current running over a shallow slope. Filling box models are often, however, based on the dynamics of vertically falling, unconfined, plumes which entrain fluid by a different mechanism to gravity currents on shallow slopes. Laboratory tank experiments of a filling box fed by a gravity current running over a shallow slope were carried out using a dye attenuation technique to investigate the development of the stratification of the ambient. These results demonstrate the differences in the stratification generated by a gravity current compared to that generated by a plume and demonstrate the nature of entrainment into gravity currents on shallow slopes.

11:09AM H2.00004 The hydraulics of exchange flow between adjacent confined building zones , SALEH NABI, MORRIS FLYNN, Dept. of Mechanical Engineering, Univ. of Alberta — Buoyancy-driven flow between two finite zones containing fluid of slightly different density is investigated. The two zones are connected through a common opening that spans the channel width so that a two layer exchange flow develops once it is removed. In the zone that initially contained dense fluid, a buoyant plume of light fluid mixes with the dense fluid leading, over time, to the development of a nontrivial ambient density stratification. Meanwhile, dense fluid flows as a gravity current into the zone that initially contained light fluid. This gravity current reflects from the end wall and propagates back toward the opening in the form of an internal bore. When the bore reaches the opening, the dynamics of the exchange flow (and consequently the source conditions of the buoyant plume) are substantially altered. Such dynamics are modeled by combining elements of gravity current, internal bore, plume and exchange flow theory; model predictions, such as that the density jump across the first front steadily decreases once the exchange flow becomes transient, are corroborated by salt-bath laboratory experiments. Substantially different predictions arise when either or both of the adjacent zones are assumed to be well-mixed so that no vertical gradient of density is allowed.

11:22AM H2.00005 Buoyancy driven turbulent flows over irregular rough surfaces , RAGHIB CHOWDHURY, ERIC RUIZ, KIRAN BHAGANAGAR, University of Texas, San Antonio — Density currents over irregular rough surfaces generated in lock-exchange mechanism have been simulated using direct numerical simulations. The surface roughness has been introduced using immersed boundary method. The governing Navier-stokes equations are solved in the vertical velocity-vertical vorticity formulation. The roughness has been characterized using statistical parameters. A preliminary analysis has been performed to understand the effect of surface roughness height on the evolution of the density currents generated in lock-exchange mechanism. The results have revealed density currents over rough wall travel at a lower speed compared to those over smooth wall. The front velocity and mixing have been characterized in terms of roughness parameters.

11:35AM H2.00006 Direct numerical simulations of gravity currents on a slope with sedimentation and resuspension , H.J. ANJUM, J.N. MCELWAINE, DAMTP, University of Cambridge, C.P. CAULFIELD, BP Institute & DAMTP, University of Cambridge — We report on direct numerical simulations of gravity currents on a slope with sedimentation and resuspension in two-dimensions. We use a hindered sedimentation model where the sedimentation flux is $u_s\phi(1-\alpha\phi)$, where u_s is the dilute limit sedimentation velocity, and ϕ is a (nondimensional) particle concentration. We consider in detail the two limiting cases of $\alpha = 0$ and $\alpha = 1$. On the boundaries we apply a zero flux boundary condition, which models resuspension. This means physically that any particles that are deposited are immediately re-entrained. On the upper surface entrainment of the ambient fluid can be balanced by sedimentation and since no buoyancy is lost through the bottom surface nearly steady currents are possible, although buoyancy is gradually lost through the tail. We compare our simulations with an analytic model based on equations for the total buoyancy, $B = \int \phi_V \, dV$ and the down-slope centre of mass $X = 1/B \int x\phi_V \, dV$. These require a closure relation based on the Froude number which we define using the vertical centre of mass $H = 1/B \int y\phi_V \, dV$ similarly to Anjum et al. (2012). We show that with this definition the Froude number is constant for most of the flow's evolution.

11:48AM H2.00007 Steady rotating density currents on a slope , WOOSOK MOON, GEORGY MANUCHARYAN, Yale University, FLORIAN SEVELLEC, University of Southampton, Yale University, ANDREW WELLS, University of Oxford, JIN-QIANG ZHONG, Tongji University, JOHN WETTLAUFER, Yale University, NORDITA — We consider the dynamics of steady turbulent entraining density currents on a conical sloping surface in a rotating fluid. A theoretical plume model is developed to describe both axisymmetric flow and single stream currents of finite angular extent. An analytical solution is derived for flow dominated by the initial buoyancy flux, which serves as an attractor for solutions with alternative initial conditions where the initial fluxes of mass and momentum are non-negligible. The solutions indicate that the downslope propagation of the current halts at a critical level where there is purely azimuthal flow in geostrophic balance, and the boundary layer approximation breaks down. A set of laboratory experiments are consistent with the dynamics predicted by the model, with flow approaching a critical level. Interpretation in terms of the theory yields an entrainment coefficient proportional to the inverse of the rotation rate ($E \propto 1/\Omega$, where E is the entrainment and Ω the rotation rate). Our theoretical model provides a framework for designing and interpreting laboratory studies of turbulent entrainment in rotating dense flows on slopes.

12:01PM H2.00008 Flushing of passive contaminants through ship ballast tanks , ZHIXIN QI, IAN EAMES, ALISTAIR GREIG, Department of Mechanical Engineering, University College London, EAMES RESEARCH GROUP TEAM — Ballast water is taken up in ports and carried by ships to maintain balance, trim, and structural integrity. Since the volume of ballast water taken up is significant, it is likely that aquatic species are taken into the tank and transported to another ecosystem. To prevent damage to ecosystems, the International Maritime Organization's Ballast Water Management Convention requires that ballast water tanks must be pumped through with at least three volumes, less is permissible if it can be demonstrated that at least a 95% of initial water is exchanged. The purpose of this talk is to examine how the exchange process depends on the geometrical complexity of the ballast tank and the density contrast between the initial ballast water and the water used for flushing. These physical processes are studied using a new mathematical model to describe flushing of contaminants from a ballast tank based on models of transport in multiroom buildings. A series of laboratory tests were undertaken using an acrylic model of a 2 to 20 compartmented ballast tank to examine the flushing of a dye from each compartment. The agreement between the theoretical and experimental results is good. These results are finally discussed in the context of international regulations for flushing ballast tanks.

12:14PM H2.00009 The Competition Between a Localised and Distributed Source of Buoyancy[1] , JAMIE PARTRIDGE, PAUL LINDEN, Cambridge University — We propose a new mathematical model to study the competition between localised and distributed sources of buoyancy within a naturally ventilated filling box. The main controlling parameters in this configuration are the buoyancy fluxes of the distributed and local source, specifically their ratio Ψ. The steady state dynamics of the flow are heavily dependent on this parameter. For large Ψ, where the distributed source dominates, we find the space becomes well mixed as expected if driven by an distributed source alone. Conversely, for small Ψ we find the space reaches a stable two layer stratification. This is analogous to the classical case of a purely local source but here the lower layer is buoyant compared to the ambient, due to the constant flux of buoyancy emanating from the distributed source. The ventilation flow rate, buoyancy of the layers and also the location of the interface height, which separates the two layer stratification, are obtainable from the model. To validate the theoretical model, small scale laboratory experiments were carried out. Water was used as the working medium with buoyancy being driven directly by temperature differences. Theoretical results were compared with experimental data and overall good agreement was found.

[1]A CASE award project with Arup.

Monday, November 19, 2012 10:30AM - 12:27PM –

Session H3 Multiphase Flows: Numerical Methods II 23B - Chair: Marcus Herrmann, Arizona State University

10:30AM H3.00001 Computation of dendritic crystal growth in supercooled water using a level-set method[1] , ANTONIO CRISCIONE, DANIEL KINTEA, ILIA V. ROISMAN, SUAD JAKIRLIC, Technical University of Darmstadt, ZELJKO TUKOVIC, University of Zagreb, CAMERON TROPEA, Technical University of Darmstadt — Over the last decades various computational approaches have been developed to simulate solidification and interfacial pattern formation phenomena. For the solution of the time-dependent moving boundary problem, which governs the dendritic crystal growth, the phase-field method has been usually used in simulations to avoid the difficulties of tracking a sharp boundary. In the present work a computational model is developed by using a level-set method. The relevant numerical algorithm is implemented into the open source software OpenFOAM. The heat transfer equations are solved in both the liquid and solid phase independently from each other. At the interface a Dirichlet boundary condition for the temperature field is imposed and a ghost-face method is applied to ensure accurate calculation of the normal derivative needed for the jump condition, i.e. for the interface-velocity in the normal direction. For the sake of updating the level set function a narrow-band around the interface is introduced. Within this band, whose width is temporally adjusted to the maximum curvature of the interface, the normal-to-interface velocity is appropriately expanded. The steady-state results are in agreement with the microscopic solvability theory.

[1]This work was supported by the German Scientific Foundation (DFG) in the framework of the SFB-TRR 75 collaborative research center.

10:43AM H3.00002 Direct numerical simulation of the Leidenfrost Effect , LUCIA RUEDA VILLEGAS, SÉBASTIEN TANGUY, Institut de Mécanique des Fluides de Toulouse (IMFT) — We present direct numerical simulations of the impact of a single droplet on a heated flat surface in the Leidenfrost regime. To that end, we solve the Navier-Stokes equations, the energy equation, and the species mass fraction equation. The Level Set method is used to track the liquid-gas interface motion and the Ghost Fluid Method is implemented to treat the jump conditions. To get rid of the temporal stability condition due to viscosity, an implicit temporal discretization is used. Some specific numerical methods have been developed to deal with droplet vaporization interface jump conditions. Since the vapor layer is very thin compared to the droplet size, a non-uniform structured grid strongly refined near the wall is used to capture the droplet bounce. We present numerical simulations that enable us to study accurately the bouncing dynamics by analyzing the momentum balance during the droplet bounce. Moreover, we determine from such computation the ratio of the droplet heat transfer flux by comparing the energy used for the phase change (latent heat) to the energy used for droplet heating (specific heat). We then compare the shape of the droplet during the impact with some experimental results.

10:56AM H3.00003 Gradient Augmented Level Set Reinitialization Approach , LAKSHMAN ANUMOLU, MARIO TRUJILLO, University of Wisconsin-Madison — A new reinitialization technique in the framework of augmented level set methods is proposed. The reinitialization PDE introduced by Sussman et al. to reconstruct signed distance function is reformulated into gradient augmented framework and solved using semi-Lagrangian approach. In order to address the issue of interfacial drift we introduce a hybrid strategy, in which this erroneous drift is handled well by anchoring the interface. In this approach, two regions, interfacial and non-interfacial regions are identified in the computational domain, where the level set and its gradient values are updated explicitly by locating the interface for the nodes belonging to the interfacial region. Two approaches are followed to identify the interface, of which, one uses the underlying Hermite polynomial evaluated along the characteristic curve, and the other uses the variant of Newton's method proposed by Chopp. Results show 4th and 3rd order spatial convergence rate for the level set function and its gradient respectively. Effect of temporal schemes is also studied with two temporal schemes, first order Euler and third order RK schemes. Unlike the numerical oscillations noted by Min in their work with Euler scheme, we have not noticed them in the presented hybrid scheme.

11:09AM H3.00004 A Quadrature-Free Conservative Level Set RKDG for Simulating Atomization[1] , ZECHARIAH JIBBEN, MARCUS HERRMANN, Arizona State University — We present an arbitrary high-order, quadrature-free, Runge-Kutta discontinuous Galerkin (RKDG) method for the solution of the conservative level set equation (Olsson et al., 2007), used for capturing phase interfaces in atomizing multiphase flows. Special care is taken to maintain high-order accuracy in the reinitialization equation, using appropriate slope limiters when necessary and a shared basis across cell interfaces for the diffusive flux. For efficiency, we implement the method in the context of the dual narrow band overset mesh approach of the Refined Level Set Grid method (Herrmann, 2008). The accuracy, consistency, and convergence of the resulting method is demonstrated using the method of manufactured solutions (MMS) and several standard test cases, including Zalesak's disk and columns and spheres in prescribed deformation fields. Using MMS, we demonstrate $k+1$ order spatial convergence for k-th order orthonormal Legendre polynomial basis functions. We furthermore show several orders of magnitude improvement in shape and volume errors over traditional WENO based distance function level set methods, and $k-1$ order spatial convergence of interfacial curvature using direct neighbor cells only.

[1]Supported by Stanford's 2012 CTR Summer Program and NSF grant CBET-1054272.

11:22AM H3.00005 A mass-conserving volume of fluid method for DNS of droplet-laden isotropic turbulence , ANTONINO FERRANTE, MICHAEL DODD, University of Washington, Seattle — We developed a mass-conserving wisps-free volume of fluid (VoF) method for direct numerical simulation (DNS) of droplet-laden turbulent flows. We used the continuous surface force (CSF) model to include the surface tension within a split-advection and mass-conserving VoF. The liquid-gas interface curvature is computed accurately using a variable-stencil height-function technique. We modified the sequence of the advection sweeps, and our results show that, in the case of non-zero Weber number, the algorithm is accurate and stable. We present DNS results of fully-resolved droplet-laden incompressible decaying isotropic turbulence at initial $Re_\lambda = 190$ using a computational mesh of 1024^3 grid points, droplet volume fraction 0.1 tracking the volumes of 7000 droplets of Weber number We $= 0.5$ based on the r.m.s. velocity fluctuation, droplet-to-fluid density ratio 10, and initial droplet diameter equal to the Taylor length-scale of turbulence.

11:35AM H3.00006 Effect of bubble-bubble interaction on mass transfer in bubbly flow using a multi-scale approach[1] , BAHMAN ABOULHASANZADEH, GRETAR TRYGGVASON, University of Notre Dame — Mass transfer in the liquid phase of gas-liquid multiphase flows generally takes place at a much shorter time scale than the momentum transfer and so leads to thin mass boundary layers around the bubble which is difficult to capture using direct numerical simulation (DNS). We have developed a subscale analytical approach using a Front Tracking method in which we use solution of a boundary layer equation for mass transfer on the bubble interface to transfer mass from bubble onto a regular grid and then we follow the mass in the domain using a reasonably coarse gird. This way we are able to considerably reduce the computational cost. Here we implement the method in a three-dimensional code and perform direct comparison of its results with experimental data. We show that this approach gives accurate results compared to experimental data and semi-empirical correlations. Then we use our subscale approach to study the effect of Reynolds number and void fraction on the effect of bubble-bubble interactions on the mass transfer in buoyant bubbly flows.

[1]Research supported by NSF.

11:48AM H3.00007 Homogeneous and isotropic turbulence laden with particles of different Stokes numbers , GEORGE MALLOUPPAS, BEREND VAN WACHEM, Imperial College London, WILLIAM GEORGE, Imperial College London, Exhibition Road, London SW7 2AZ — The interactions of homogeneous isotropic turbulence with particles of various Stokes numbers are examined. The talk focuses on a series of DNS of forced turbulence laden with discrete particles performed on a 128^3 periodic box. Several parameters are varied such as the Stokes number and the Taylor Reynolds number. The modification of one-point statistics due to the presence of particles is investigated. Moreover, the relation of forcing with light and heavy particles is investigated by evaluating the correlation coefficients between the forcing and the particles. It is shown that our newly proposed forcing scheme has a limited direct effect on the particles. An important consequence of the presence of the particles is the modification of the dissipation spectrum of the fluid, which depends on the particle Stokes number, and more surprisingly, on the Taylor Reynolds number. This is examined in view of the two-way coupling spectrum, which acts as a dissipative-transfer term. The dispersion of fluid and discrete particles are compared with analytical solutions by assuming the velocity autocorrelation function is of exponential form. Finally, the talk will address the importance of preferential concentration of the particles and its effect on the two-way coupling.

12:01PM H3.00008 On the direct numerical simulation of moderate-Stokes-number turbulent particulate flows using algebraic-closure-based and kinetic-based moments methods , AYMERIC VIE, EM2C/Ecole Centrale Paris, ENRICA MASI, OLIVIER SIMONIN, IMFT, MARC MASSOT, EM2C/Ecole Centrale Paris, EM2C/ECOLE CENTRALE PARIS TEAM, IMFT TEAM — To simulate particulate flows, a convenient formalism for HPC is to use Eulerian moment methods, which describe the evolution of velocity moments instead of tracking directly the number density function (NDF) of the droplets. By using a conditional PDF approach, the Mesoscopic Eulerian Formalism (MEF) of Février et al. 2005 offers a solution for the direct numerical simulation of turbulent particulate flows, even at relatively high Stokes number. Here, we propose to compare to existing approaches used to solved for this formalism: the Algebraic-Closure-Based Moment method (Kaufmann et al. 2008, Masi et al. 2011), and the Kinetic-Based Moment Method (Yuan et al. 2010, Chalons et al. 2010, Vié et al. 2012). Therefore, the goal of the current work is to evaluate both strategies in turbulent test cases. For the ACBMM, viscosity-type and non-linear closures are envisaged, whereas for the KBMM, isotropic and anisotropic closures are investigated. A main aspect of the current methodology for the comparison is that the same numerical methods are used for both approaches. Results show that the new non-linear closure and the Anisotropic Gaussian closures are both accurate in shear flows, whereas viscosity-type and isotropic closures lead to wrong results.

12:14PM H3.00009 Simulating Primary Atomization at Arbitrary Density Ratios: a Stable and Conservative Framework , VINCENT LE CHENADEC, Stanford University, HEINZ PITSCH, Institute for Combustion Technology, RWTH Aachen — The present work focuses on two recent developments for Direct Numerical Simulation of two-phase flows, and their application to computations of turbulent primary atomization of liquid jets at large density ratios. Mass conservation properties of the algorithm are improved by means of a second-order unsplit Volume-of-Fluid method coupled to the Level Set approach. The three-dimensional volume fraction transport scheme is shown to reduce numerical artifacts known to pollute the interface representation in under-resolved regions of the flow. In the interface vicinity, the momentum conservation as well as stability of the flow solver are guaranteed by a monotonicity preserving geometric transport of the momentum, defined consistently with the volume fraction transport. Away from the interface, the flux computation is switched to a centered discretization in order to avoid excessive numerical dissipation. This framework is assessed in a set of validation cases, and applied to simulate the primary atomization of a turbulent round jet in quiescent gas at air/water density ratio and moderate Reynolds and Weber numbers.

Monday, November 19, 2012 10:30AM - 12:40PM –

Session H4 Drops VI 23C - Chair: Stephane Zaleski, Institut D039;Alembert, CNRS and UPMC

10:30AM H4.00001 Effect of the liquid/solid friction on the drop impact dynamics , CHRISTOPHE PIRAT, HENRI LASTAKOWSKI, ANNE-LAURE BIANCE, CHRISTOPHE YBERT, LPMCN UCBL UNIV. LYON 1, LIQUID AT INTERFACES TEAM — In this experimental study, the problem of drop impact on a smooth solid surface is investigated, both in wetting and non-wetting configurations. The combination of high speed Particle Image Velocimerty and absorption method allows us to probe the flow in the lamella during an impact. The role played by the solid/liquid friction in the sheet spreading is clarified by comparing impacts in low friction (above the Leidenfrost temperature) and partial wetting (on a cold sustrate). In the latter case, the results show that the lamella reaches a thickness limited by the development of a viscous boundary layer, in very good agreement with recent theoretical and numerical results. The boundary layer is not observed in the low friction case.

10:43AM H4.00002 An aerodynamical mechanism for droplet splashing , CHRISTOPHE JOSSERAND, ZHEN JIAN, Institut D'Alembert, CNRS & UPMC, STEPHANE POPINET, NIWA, New-Zealan, PASCAL RAY, STEPHANE ZALESKI, Institut D'Alembert, CNRS & UPMC — Using a numerical study of drop impact on a solid substrate, we propose an aerodynamical mechanism to explain the dependance of the impact on the gas properties. Varying both the viscous and density ratio between the drop liquid and the external gas in the incompressible limit, we describe the frontier between spreading and splashing. Although a jet is sometimes formed before contact, we show that the splashing dynamics is controlled by the aerodynamical pressure exerted by the gas on the expanding liquid sheet.

10:56AM H4.00003 Splash Criteria for Liquid Drop Impact on Smooth, Dry Surfaces , CACEY STEVENS, SIDNEY NAGEL, The University of Chicago — It is important to find a criterion that predicts the transition from smooth deposition to splashing of low-viscosity liquid drops when they land on a smooth, dry surface. Using high-speed imaging, we have determined the threshold pressure, P_T, of the ambient gas for which a liquid drop splashes as a function of the relevant parameters (gas molecular weight m_G, liquid viscosity ν_L, surface tension σ, drop diameter D, and impact speed U_0). There is a non-monotonic trend of P_T versus U_0 [1]. We find this same trend as we systematically change other liquid and gas properties; they simply shift the curve. By defining a scaled pressure, $P_{T,scaled} = P_T D^{0.5} \nu_L^{0.25} m_G^{0.5} \sigma^{-0.25}$, and scaled impact speed $U_{0,scaled} = U_0 D^{0.3} \nu_L^{0.4} \sigma^{-0.35}$, we find a collapse of all data sets onto a single curve. This scaling applies to both high and low velocity regimes.

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

11:09AM H4.00004 Substrate topography-mediated air film collapse below an impacting drop , JOLET DE RUITER, DIRK VAN DEN ENDE, FRIEDER MUGELE, University of Twente — Liquid drops hitting solid surfaces are slowed down by the ambient air layer that needs to be squeezed out before the liquid actually touches the solid. How does substrate topography mediate the collapse of the air film? For moderate velocity impacts (Weber number around unity) we show that a drop gently spreads on an undulated air layer that thins to a minimum of several hundreds of nanometers. Whether or not the air layer collapses (in a single spot nucleation) leading to wetting of the substrate, can be tuned with substrate topography. Introducing micro- and nanoscale defects of various topography, we observe the two different cases, i.e. drop bouncing on the air layer and directed nucleation of liquid-solid contact. Using quantitative dual wavelength reflection interference microscopy we reveal the evolution of the air film, and the influence of substrate defects on localized pressure build-up and film collapse.

11:22AM H4.00005 Drop impact of suspensions , FRANCOIS BOYER, JACCO SNOEIJER, FRITS DIJKSMAN, DETLEF LOHSE, Physics of Fluids, University of Twente — Drop impact dynamics on solid surfaces is a classical subject of interfacial hydrodynamics, which occurs in many industrial and environmental situations. So far, most of the studies have concerned Newtonian fluids. Complex fluids (polymer dispersions, particle suspensions, gels, emulsions, ...) are however of considerable interest for a wide range of applications. From a fundamental point of view, how the non-Newtonian features of a complex liquid change (drastically in some cases) the drop impact dynamics is a challenging open problem. Newly observed phenomena will then be presented.

11:35AM H4.00006 Dynamic Leidenfrost temperature for impact of droplets on micro-structured surfaces , HENDRIK J.J. STAAT, TUAN TRAN, Physics of Fluids Group, Faculty of Science and Technology, University of Twente, 7500AE Enschede, The Netherlands, ARTURO SUSARREY ARCE, Catalytic Processes and Materials Group, Faculty of Science and Technology, University of Twente, 7500AE Enschede, The Netherlands, TOBIAS C. FOERTSCH, Physics of Fluids Group, Faculty of Science and Technology, University of Twente, 7500AE Enschede, The Netherlands, ARIE VAN HOUSELT, Catalytic Processes and Materials Group, Faculty of Science and Technology, University of Twente, 7500AE Enschede, The Netherlands, HAN GARDENIERS, Mesoscale Chemical Systems Group, Faculty of Science and Technology, University of Twente, 7500AE Enschede, The Netherlands, ANDREA PROSPERETTI, DETLEF LOHSE, CHAO SUN, Physics of Fluids Group, Faculty of Science and Technology, University of Twente, 7500AE Enschede, The Netherlands — When a droplet impacts a surface heated above the liquid's boiling point, the droplet either contacts the surface and boils immediately, (contact boiling) or is supported by a developing vapor layer and bounces back (film boiling & Leidenfrost state). We study the transition between these two different behaviors and how it is affected by the control parameters such as impact velocity and controlled roughness (i.e., micro-structures fabricated on silicone surfaces). Additionally, in the film boiling regime, we show that the resident time of the droplet impact is insensitive to the velocity, the surface temperature, and the structure's geometry. In the contact boiling regime, we show that the structured surfaces induce the formation of liquid jets emerging during the spreading stage of the impacting droplets.

11:48AM H4.00007 Impact of droplet on superheated surfaces , DETLEF LOHSE, HENDRIK J.J. STAAT, TUAN TRAN, University of Twente, ANDREA PROSPERETTI, University of Twente and Johns Hopkins University, CHAO SUN, University of Twente — At impact of a liquid droplet on a smooth surface heated way above the liquid's boiling point, the droplet spreads without any surface contact, floating on its own (Leidenfrost-type) vapor layer, and then bounces back. We show that the dimensionless maximum spreading factor Γ, defined by the ratio of the maximal spreading diameter and the droplet diameter, shows a universal scaling $\Gamma \sim We^{\gamma}$ with the Weber number We – regardless of surface temperature and of liquid properties – which is much steeper than that for the impact on non-heated (hydrophilic or hydrophobic) surfaces, for which $\gamma = 1/4$. Based on the idea that the vapor shooting out of the gap between the droplet and the superheated surface drags the liquid outwards, we derive scaling laws for the spreading factor Γ, the vapor layer thickness, and the vapor flow velocity.

12:01PM H4.00008 Microdroplet impact at very high velocity[1] , CLAAS WILLEM VISSER, YOSHIYUKI TAGAWA, CHAO SUN, DETLEF LOHSE, Universiteit Twente, PHYSICS OF FLUIDS GROUP TEAM — At APS-DFD 2011, we presented preliminary data of water microdroplet impact at velocities up to 100 m/s and droplet diameters from 12 to 100 μm. Now we place these results into context and use them to improve understanding of droplet spreading. The parameter range covers the transition from capillary-limited to viscosity-limited spreading of the impacting droplet. The maximum spreading radius is compared to several existing models. The model by Pasandideh-Fard et al. (1996) agrees well with the measured data, indicating the importance of a thin boundary layer just above the surface. Here, most of the viscous dissipation of the spreading droplet takes place. As explained by the initial air layer under the impacting droplet, a contact angle of 180 degrees is used as model input.

[1]Stichting Fundamenteel Onderzoek der Materie is acknowledged for funding

12:14PM H4.00009 How Does Air Evolve into a Bubble During Drop Impact? , JI SAN LEE, BYUNG MOOK WEON, SU JI PARK, JI TAE KIM, JAEYEON PYO, JUNG HO JE, Pohang University of Science and Technology, KAMEL FEZZAA, X-ray Science Division, Advanced Photon Source, Argonne National Laboratory, X-RAY IMAGING CENTER TEAM — When a liquid drop impacts on a solid substrate, a tiny air film is generally entrapped between the drop and the substrate and eventually evolves into a bubble by surface energy minimization. We investigated how air evolves into a bubble during drop impact using ultrafast x-ray phase-contrast imaging that enables us to track the detailed morphological changes of air with high temporal and spatial resolutions. We found that the evolution takes place through complicated three stages: inertial retraction of the air film, collapse of the top air surface into a bubble, and pinch-off of a daughter droplet in the bubble. The collapse and the pinch-off can be explained by energy convergence that is associated with Ohnesorge number (Oh) regarding capillary waves and viscous damping. We measured a critical Oh number, Oh* $\sim 0.026 \pm 0.003$, above which the generation of the daughter droplet is suppressed. Interestingly we found that the bubble is detached favorably from wettable surfaces, which suggests a feasible way to eliminate bubbles for many applications by controlling surface wettability. The threshold angle for bubble detachment was measured as $\sim 40 \pm 5$ deg. for water, which agrees with a geometrical estimation.

12:27PM H4.00010 Pinning of a perfectly wetting volatile liquid at a sharp edge - experiment and theory[1] , YANNIS TSOUMPAS, SAM DEHAECK, ALEXEY REDNIKOV, Universite Libre de Bruxelles - TIPs, CP 165/67, MARIANO GALVAGNO, UWE THIELE, Loughborough University - Department of Mathematical Sciences, PIERRE COLINET, Universite Libre de Bruxelles - TIPs, CP 165/67 — It is well known that contact lines of drops, even those of wetting liquids, stay pinned at sharp edges of the substrate until the apparent contact angle exceeds a critical value. In the present study, we show that evaporation influences this effect. The edge of a circular groove is used as an example. Experiments with wetting liquids of different evaporation rate show indeed that not only the spreading of the liquid is adequately halted, but also that the pinning is enhanced for the more volatile cases. The experimental results are qualitatively compared with predictions of a thin film model in two dimensions. The approach employs an evolution equation for the height profile of an evaporating thin film (small contact angle droplet) on a substrate with a rounded edge, and enables one to predict the dependence of the apparent contact angle on the position of the contact line. The calculations confirm our experimental observations, namely that there exists a dynamically produced critical angle for depinning that increases with the evaporation rate. This suggests that one may introduce a simple modification of the Gibbs criterion for pinning that accounts for the non-equilibrium effect of evaporation.

[1]Supported by the Marie Curie MULTIFLOW Network, by ESA & BELSPO, and by FRS-FNRS.

Monday, November 19, 2012 10:30AM - 12:27PM –

Session H5 Computational Fluid Dynamics V 24A - Chair: Sourabh Apte, Oregon State University

10:30AM H5.00001 Multi-scale Numerical Simulations of Magnetic Fluids[1] , PHILIP YECKO, Montclair State University, New Jersey, RUBEN SCARDOVELLI, Universita di Bologna, Italy, HOLLY TIMME, Virginia Tech, A. DAVID TRUBATCH, Montclair State University, New Jersey — We develop, validate and apply a new Height Function (HF) based Volume of Fluid (VOF) code to the simulation of ordinary and magnetic fluids in two-dimensional and three-dimensional axisymmetric geometries. The HF algorithm provides second-order accurate curvature and interface normal formulation, improving surface tension and magnetic stress accuracy. Motivated by our recent experimental results on ferrofluid rheology dominated by field-induced magnetic particle threads we have applied this code to the meso-scale problem of the interaction of flow with threads. The multiscale approach allows us to model solid particles embedded in bulk fluid, approximating the magnetic particle threads –which are several microns (thousands of magnetic nanoparticles) wide– by means of a chain of several (from 3 to 11) pseudo-particles with magnetic properties intermediate between the ferrofluid and the magnetic particles. In the presence of an imposed uniform field, these pseudo-particles form chains (threads) along the field direction. By placing a single pseudo-particle chain in a uniform flow, simple shear flow and/or a pure straining flow we can directly examine the equilibrium chain orientation and hydrodynamic stresses which characterize the interaction between thread and shear. Our simulation results allow us to compute the energy dissipation and thus model the enhanced drag effected by a thread on fluid flow.

[1]We acknowledge support by N.S.F. Grant No. 1016383

10:43AM H5.00002 LES of turbulent boundary layer flow over irregular and multiscale topographies, and comparison with experimental data , WILLIAM ANDERSON, Mechanical Engineering Dept., Baylor University, KENNETH CHRISTENSEN, Department of Mechanical Science and Engineering, University of Illinois at Urbana-Champaign — Topographies featuring an irregular distribution of obstacles occur frequently in fluid machinery applications. Moreover, the distribution of characteristic size of these obstacles may also be broad. The hydrodynamic response of turbulence to such topographies is complicated, since both the flow and topography are composed of multiple length scales and identification of dominant scale is not obvious. Mejia-Alvarez and Christensen, 2010: Phys. Fluids, 22 015106 presented comprehensive experiments of developing and developed turbulent boundary layer flow over multiscale gas turbine blade topography. In the present work, results of large-eddy simulation of turbulent boundary layer flow over the topography considered by the experimentalists is presented. The topography is resolved with an immersed boundary method. The numerical and experimental results show reasonable agreement. The results are also used to make a posteriori evaluation of parameters used in classical relations linking the hydrodynamic roughness length to other statistics of the topography.

10:56AM H5.00003 Numerical Study of Nusselt Number in a Heated Pipe With the Use of Variable-Order Resolution , KENNETH DAVIS, Department of Mechanical Engineering and Materials Science, Rice University, PAUL FISCHER, Mathematics and Computer Science Division, Argonne National Laboratory — We present results for a numerical study of turbulent heat transfer in a pipe with a constant heat flux at the wall. Nusselt numbers are computed for Reynolds numbers between 5,000 and 15,000 over a wide range of Prandtl numbers and the results are compared to the Dittus-Boelter relation. The simulations are based on the spectral element method in which the velocity and pressure are represented by tensor-product polynomials of degree N in each of E elements. Typical values of N are in the range 4 to 20 and, for this study, E is between 4000 and 12000. We examine potential savings of using elevated resolution for the temperature field only, which is particularly interesting for the case $Pr > 1$. Specifically, for water flow, one has as Peclet to Reynolds number ratio of approximately six, which implies a need for elevated resolution of the temperature field. This study explores the relative convergence rates and costs when the polynomial order for the temperature field, N_t, is increased with respect to N. We identify the optimal ratio, N_t/N, as a function of Prandtl number as grid convergence is attained.

11:09AM H5.00004 Parallel performance of iterative Poisson Solvers for Uniform and Structured Adaptive Grids on a Cray XT5 (Kraken)[1] , MARCOS VANELLA, ELIAS BALARAS, The George Washington University — The Poisson equation solution is of great importance in areas as diverse as Gravitation and Electrostatics to fractional step methods for viscous incompressible flows. For uniform grids fast direct methods based on FFT are usually adopted, which scale well to thousands of processors. For block-wise structured adaptive grids, scaling depends on the amount of communication and load balancing inherent on the solution method. In the present study we report scaling tests on uniform and two-level grids of increasing size. A direct solver based on trigonometric transforms is used on the uniform grid cases, and three different iterative solvers are used on the adaptive cases. In particular, two multigrid algorithms and a bi-conjugate gradient stabilized algorithm, (BiPCGSTAB), preconditioned by one sweep of the Multigrid (no corrections applied in preconditioned) are utilized. All solvers are implemented within the Flash software infrastructure. The computations are performed on the Kraken, Cray XT5 supercomputer employing up to 40,000 cores. It is seen that interprocessor communication required by the data redistribution operations, and the base level fast solution has an important effect on the decline in parallel scaling.

[1]We acknowledge the financial support from National Science Foundation (OCI-0904920). Computing time was provided by XSEDE

11:22AM H5.00005 Comparative Efficiency of Implicit, Explicit and Implicit-Explicit Strong Stability Preserving Methods , GABRIEL MORAES, RENAN TEIXEIRA, Instituto Militar de Engenharia, LEONARDO ALVES, Universidade Federal Fluminense — Several unsteady problems in transport phenomena require highly accurate solutions. Attempts to increase accuracy order of most numerical schemes, however, often weaken their linear and/or nonlinear stability. On the other hand, Strong Stability Preserving (SSP) methods are able to increase their accuracy order in time while still maintaining the overall stability properties of the original forward Euler method from which they were generated. This is achieved by, among other things, restricting the maximum allowed time step of these schemes. Hence, most recent studies have focused on developing optimal SSP schemes, i.e., minimally restrictive. Under this context, a significant variety of explicit and implicit formulations for multi-step and multi-stage marching schemes of different accuracy orders has been created. Despite their popularity, some open issues still remain. Linear stability of implicit schemes usually allows very large time steps, making them cost effective for many applications when compared to their explicit counterparts. Hence, the SSP time step restriction might render these schemes comparatively inefficient. The present study evaluates different explicit, implicit and implicit–explicit SSP time integration schemes for a series of test cases in an attempt to distinguish the most efficient scheme according to an error / computer time analysis. All three schemes employed a second order TVD flux limiter discretization for the spatial derivatives.

11:35AM H5.00006 Superimposition of external oscillation to enhance heat transfer from objects in cross flow , RAED BOURISLI, Kuwait University — Laminar flow around objects gives rise to the recurrent build-up and release of vortices on alternate sides of the objects over a wide range of Reynolds numbers. Inherent disturbance of the otherwise uniform flow and temperature fields plays an important role in many structural, hydrodynamic as well as thermal aspects of situations where it is present. For example, the local disturbance of the velocity field leads to subsequent instability in the temperature field, causing variations in local Nusselt number, heat flux and surface temperature, among other things. One can take advantage of this phenomenon in many applications such as the cooling of electronic equipment. It is suggested here that the intensity of the outlined vortex shedding phenomenon can be deepened if an external movement is superimposed on the velocity of the structure or any nearby object. Numerical test of several objects rotated in-plane: cylinders, squares, triangles and horizontal plates, are performed. The key physical observation is the relative magnitudes of the heat transfer due to natural vortex shedding compared to the added value obtained by superimposing an additional external source of oscillation. A realistic case of electronics chips cooling is presented to show the effect of matching the natural frequency of vortex shedding by that of the inhomogeneity (Video: 0-9 s optimum; 9-18 non-). In this case, vortex shedding from the plates plays a smaller role in disturbing the flow, hindering it at times. When the two frequencies coincide, however, in-phase shedding leads to more efficient heat transfer.

11:48AM H5.00007 Statistic fluid dynamic of multiphase flow[1] , HYUNKYUNG LIM, JAMES GLIMM, YIJIE ZHOU, XIANGMIN JIAO, Stony Brook University — We study a turbulent two-phase fluid mixing problem from a statistical point of view. The test problem is high speed turbulent two-phase Taylor-Couette flow. We find extensive mixing in a transient state between an initial unstable and a final stable configuration. With chemical processing as a motivation, we estimate statistically surface area, droplet size distribution and transient droplet duration.

[1]This work is supported in part by the Nuclear Energy University Program of the Department of Energy, Battelle Energy Alliance LLC 00088495.

12:01PM H5.00008 Comparative Study of Reynolds Averaged and Embedded Large Eddy Simulations of a High Pressure Turbine Stage[1] , ALEKSANDAR JEMCOV, THEODORE WILLIAMS, THOMAS CORKE, University of Notre Dame — An Embedded Large Eddy Simulation (ELES) approach is used to simulate the flow path through a high pressure turbine stage. The turbine stage includes the entry duct, stationary inlet and exit guide vanes, and a rotor. The rotor blade design includes a squealer tip. The flowfield around the rotor is simulated using LES. A Reynolds Averaged Simulation (RAS) is used to simulate the rest of the flow domain. The interface between RAS and LES domains uses the RAS turbulence quantities as a means of obtaining length scales that are used in computing the vorticity that is required to trigger a proper energy cascade within the LES part of the flow field. The ELES approach allows for substantial computational savings since it allows for different mesh resolutions in various parts of the computational domain as needed. The objective of this work is to observe at a lower computational cost, the local flow features that cannot be resolved in a RAS approach. A comparative analysis between RAS and ELES approaches for this turbomachinery problem is then presented.

[1]Supported by NASA

12:14PM H5.00009 ABSTRACT WITHDRAWN –

Monday, November 19, 2012 10:30AM - 12:27PM –

Session H6 Microfluidics: General II 24B - Chair: David Saintillan, University of Illinois at Urbana-Champaign

10:30AM H6.00001 Development of a modified Hess-Murray law for non-Newtonian fluids in bifurcating micro-channels[1] , DAVID EMERSON, ROBERT BARBER, STFC Daresbury Laboratory — Microfluidic manifolds frequently require the use of bifurcating channels and these can be used to create precise concentration gradients for chemical applications. More recently, novel devices have been attempting to replicate vasculatures or bronchial structures occurring in nature with the goal of creating artificial bifurcations that mimic the basic principles of designs found in nature. In previous work, we have used the biological principles behind the Hess-Murray Law, where bifurcating structures exhibit a constant stress profile and follow a third-power rule, to enable rectangular or trapezoidal micro-channels to be fabricated using conventional lithographic or wet-etching techniques. Using biological principles to design man made devices is generally referred to as biomimetics and this approach has found success in a range of new and emerging topics. However, our previous work was limited to Newtonian flows. More recently, we have used the Rabinovitsch-Mooney equation to be able to extend our analysis to non-Newtonian fluids. This has allowed us to develop a new rule that can provide a design criterion to predict channel dimensions for non-Newtonian flows obeying a constant stress biological principle.

[1]The Engineering and Physical Sciences Research Council for support of CCP12 and Programme Grant award (grant number EP/I011927/1)

10:43AM H6.00002 Fluidic Control by Capillary and Maxwell Stresses for Liquid Printing of Small Metallic Structures , GERRY DELLA ROCCA, SANDRA TROIAN, California Institute of Technology, MC 128-95, Pasadena, CA 91125 — Liquid dosing strategies for microfluidic applications normally rely on interior flow driven by external pressure gradients. To maintain a constant flow rate, the effective pressure drop over a given length conduit must scale inversely as the fourth power in the conduit radius, as prescribed by the Hagen-Poiseuille relation. For micron scale capillaries, this constraint requires enormous pressure gradients enforced by large pumps, cascades of tubing material, and electronic sensors. This burden, coupled with the likelihood of occlusions due to gas bubbles, contaminant or carrier particles, limits the usefulness of enclosed flow for transporting very small fluid volumes. Capillary flow on substrates etched with slender open grooves provides a much simpler, less expensive, efficient and reliable method of transport. When coupled with flow modulation by remote electric fields, the flow can be metered effectively and much more rapidly. We discuss the steady state, transient and oscillatory flow of a perfectly conducting liquid within an open conduit subject to a spatially and temporally varying electric field. The geometry investigated is geared toward applications involving liquid printing of small metallic elements for large-area circuits and photovoltaic displays.

10:56AM H6.00003 The hydrodynamic interaction between a soft particle and a permeable surface , GUY RAMON, Princeton University, HERBERT HUPPERT, University of Cambridge, HOWARD STONE, Princeton University — Practical experience has shown that permeable surfaces are more prone to deposition and, consequently, foul more than other non-permeable surfaces. This is due to the presence of an additional velocity component perpendicular to the surface. A particle will translate towards the surface at the same velocity as the background fluid; however, at close approach, particle interaction with the surface creates additional forces resulting from electrostatic, dispersion, polar and, the focus of this work, hydrodynamic interactions. A lubrication approximation is used to derive an equation for the pressure field; coupling with the elastic response of the particle allows evaluation of elastic interaction when the particle and/or surface are not completely rigid (e.g., soft polymer interfaces, bacteria cells, etc.). Useful asymptotic forms are derived, offering a clear and intuitive understanding of the force acting on a particle at close approach to a surface and its dependence on particle size, shape, the background flow and permeability of the surface.

11:09AM H6.00004 Numerical analysis of radiation- and streaming-induced microparticle acoustophoresis[1] , RUNE BARNKOB, PETER BARKHOLT MULLER, HENRIK BRUUS, Technical University of Denmark, MADS JAKOB HERRING JENSEN, COMSOL A/S — We present a numerical analysis of the acoustophoretic motion of microparticles suspended in a liquid-filled microchannel excited with an ultrasound field tuned to resonance. The imposed first-order ultrasound field generates second-order fields leading to two particle forces with a non-zero time-average: the acoustic radiation force from sound-wave scattering off the particles and the Stokes drag force from the induced acoustic streaming flow. We consider a viscous heat-conducting liquid and non-interacting spherical particles. The model is based on the thermoviscous acoustic equations and takes into account the micrometer-thin but crucial viscous boundary layers at rigid walls. Using a numerical tracking scheme, we quantify the acoustophoretic particle velocities for experimentally relevant parameters. We characterize the transition from radiation- to streaming-dominated acoustophoretic motion as function of particle size, channel geometry, and material properties. See also Muller *et al.*, Lab Chip **12**, in press (2012).

[1]This research was supported by the Danish Council for Independent Research, Technology and Production Sciences, Grants No. 274-09-0342 and No. 11-107021.

11:22AM H6.00005 Experimental analysis of radiation- and streaming-induced microparticle acoustophoresis , MASSIMILIANO ROSSI, ALVARO MARIN, CHRISTIAN J. KÄHLER, Bundeswehr University Munich, PER AUGUSTSSON, THOMAS LAURELL, Lund University, PETER B. MULLER, RUNE BARNKOB, HENRIK BRUUS, Technical University of Denmark — We present an experimental analysis of the acoustophoretic motion of microparticles suspended in a liquid-filled acoustofluidic microchannel. This analysis intends to provide an experimental validation and support to very recent numerical and analytical models of radiation- and streaming-induced microparticle acoustophoresis (see Muller et al., Lab Chip 12, in press, 2012). For the experiments, we used a suspension of water and spherical polystyrene particles in a straight microchannel with rectangular cross section, actuated in its 1.94-MHz resonance by means of a piezoelectric transducer. The particles were labeled with a fluorescent dye and their motion was observed using an epifluorescent microscope. For the analysis, the Astigmatism Particle Tracking Velocimetry (APTV) technique was used to measure the three-dimensional trajectories and velocities of the particles with high precision and resolution (Cierpka et al., Meas Sci Technol 22, 2011). The experiments were performed for different particle sizes, ranging from 0.5-μm particles, dominated by the Stokes drag force induced by the acoustic streaming of the flow, to 5-μm particles, dominated by the acoustic radiation force. The results agree well with the analytical and numerical predictions.

11:35AM H6.00006 Inertial particle trapping and transport in viscous streaming , KWITAE CHONG, JEFF D. ELDREDGE, University of California, Los Angeles — A probe undergoing rectilinear oscillation creates a steady large-scale circulatory flow, which is conventionally called viscous streaming. This streaming flow, generated by the nonlinear interaction of the primary oscillatory motion, can provide an appealing option in micromanipulation, such as trapping, positioning and transport of a discrete particle. In this study, the streaming flow around a circular cylinder is obtained from previous analytical solution (by asymptotic expansion in small amplitude). The motion of an inertial particle in this flow is obtained by integrating the Maxey-Riley equation, in which the wall effect is newly considered. It has been observed in our previous work that, under certain conditions, the inertial particle is trapped inside the center of a streaming cell near the probe; here, the manner of trapping is re-explored under various choices of physical parameters, such as Reynolds number, particle size and density. We also extend the study to various arrangements of multiple oscillating probes by using high-fidelity computations to simulate particle transport between probes. In particular, we demonstrate systematic particle transport between probes by selectively stopping and starting the oscillatory motion of adjacent probes.

11:48AM H6.00007 Drops subjected to surface acoustic waves: flow dynamics , PHILIPPE BRUNET, CNRS - Laboratoire Matière et Systèmes Complexes (MSC), MICHAEL BAUDOIN, Institut d'Electronique de Microélectronique et Nanotechnologies (IEMN) - Université Lille 1, OLIVIER BOU MATAR, Institut d'Electronique de Microélectronique et Nanotechnologies (IEMN) -Ecole Centrale de Lille, DYNAMIQUE DES SYSTÈMES HORS EQUILIBRE TEAM, AIMAN - FILMS TEAM — Ultrasonic acoustic waves of frequency beyond the MHz are known to induce streaming flow in fluids that can be suitable to perform elementary operations in microfluidics systems. One of the currently appealing geometry is that of a sessile drop subjected to surface acoustic waves (SAW). Such Rayleigh waves produce non-trival actuation in the drop leading to internal flow, drop displacement, free-surface oscillations and atomization. We recently carried out experiments and numerical simulations that allowed to better understand the underlying physical mechanisms that couple acoustic propagation and fluid actuation. We varied the frequency and amplitude of actuation, as well as the properties of the fluid, and we measured the effects of these parameters on the dynamics of the flow. We compared these results to finite-elements numerical simulations.

12:01PM H6.00008 Surface Acoustic Wave (SAW) based Microfluidics for Particle and Droplet Manipulation , YE AI, BABETTA L. MARRONE, Advanced Measurement Science, Los Alamos National Laboratory, Los Alamos, New Mexico 87545, USA — Acoustics has emerged as one of the most promising non-invasive techniques for particle and droplet manipulation in microfluidics. Surface acoustic wave (SAW) based microfluidic devices are developed to manipulate micron-sized particles and discrete droplets. When solid particles are immersed in a standing SAW, the resulting acoustic radiation force acting on the particles can drive the particles into the pressure node, resulting in particle focusing phenomenon. The amplitude of the acoustic radiation force highly depends on the particle properties, leading to different acoustic responses for different types of particles. Separation of two types of fluorescent particles is demonstrated using the developed SAW-based microfluidic device. Numerical simulations are performed to study the generation of the standing SAW and the particle separation which is in good agreement with the experimental results. When a SAW propagates through a droplet in contact with the piezoelectric substrate, the SAW partially leaks into the droplet and exerts an acoustic streaming force in the droplet, which can move the droplet in the direction of SAW propagation. It is further found that a curved SAW transducer is able to focus SAW into a very narrow beam and in turn focus randomly distributed droplets into a specific target. It is demonstrated that focused SAWs can be more efficient than uniform SAWs for droplet actuation in microfluidics.

12:14PM H6.00009 Buckling and Transport of Semiflexible Filaments in Cellular Flows , HARISHANKAR MANIKANTAN, DAVID SAINTILLAN, University of Illinois at Urbana-Champaign — A slender elastic filament placed in a lattice of counter-rotating vortices is known to move as a random walker. Such a cellular flow has also been compared to experiments on actin transport across myosin beds. We present numerical results that for the first time include the effect of Brownian fluctuations on these transport properties. A semiflexible filament is modeled based on slender-body theory for Stokes flow, and incorporates Euler-Bernoulli elasticity as well as thermal fluctuations. We consider inextensible biopolymers of length of the order of persistence length (actin, microtubules etc). In a hyperbolic external flow, such an elastic filament is susceptible to a buckling instability that drives it between stagnation points in the lattice. The velocity distribution of the filament is bimodal in the non-Brownian case, and systematically flattens out with thermal fluctuations. Also, filaments are shown to spend time waiting in a cell before being pushed out by a random fluctuation. Such a waiting time distribution might indicate sub-diffusive transport as against diffusive transport seen in the non-Brownian case. We also study the distribution of mass of the filament across the lattice, and discuss how persistence length affects its preferred position in a unit cell.

Monday, November 19, 2012 10:30AM - 12:40PM –

Session H7 Non-Newtonian Flows I 24C - Chair: Anette Hosoi, Massachusetts Institute of Technology

10:30AM H7.00001 Flow structure of polymer solutions close to walls, at less than one correlation length , PATRICK TABELING, ZHENZHEN LI, MARC YONGER, FABRICE MONTI, EMMANUEL TERRIAC, CHOONGYEOP LEE, Microfluidics MEMS and Nanostructures Laboratory - ESPCI, MICROFLUIDICS MEMS AND NANOSTRUCTURES LABORATORY - ESPCI TEAM — We measure flow profiles in semi-dilute PEO solutions at distances less than a correlation length, close to smooth walls, using an evanescent wave technique we have been developing in the lab for a number of years. We observed unexpected flow structures in the first two correlation lengths from the wall, albeit consistent with recent numerical studies.

10:43AM H7.00002 Interface Oscillation in the Side-by-Side (SBS) Tape Casting of Functionally Graded Ceramics (FGCs) , MASOUD JABBARI, Department of Mechanical Engineering, Technical University of Denmark, REGINA BULATOVA, Department of Energy Conversion and Storage, Technical University of Denmark, JESPER HATTEL, Department of Mechanical Engineering, Technical University of Denmark, CHRISTIAN BAHL, Department of Energy Conversion and Storage, Technical University of Denmark — Room temperature magnetic refrigeration is a new highly efficient and environmentally protective technology. Although it has not been maturely developed, it shows great applicable prosperity and seems to be a potential substitute for the traditional vapor compression technology. Tape Casting is a common process in producing multilayer ceramics, which now is used for producing side-by-side (SBS) functionally graded ceramics (FGCs). These FGCs are mostly used in the magnetic refrigeration sectors due to the varying composition of the magnetocaloric materials so that the magnetic transition temperature of the magnetic regenerator varies along the paths. The main goal of this research is to study the multiple material flow in SBS tape casting and analyze its influence on the interface between the stripes. The materials used for the experimental part are $La_{0.85}Sr_{0.15}MnO_3$ and $Ce_{0.9}Gd_{0.1}O_2$ ceramic slurries. The rheological behavior of the slurries are extracted from experiments and used in the ANSYS FLUENT commercial code to develop a fluid flow model for the non-Newtonian ceramic slurries and evaluate the interface oscillation between the stripes in SBS tape casting. The Numerical results show reasonable agreement with corresponding experimental results.

10:56AM H7.00003 Surface Characterization of pNIPAM Under Varying Absolute Humidity , ARNAV CHHABRA, RAVITEJ KANAPURAM, HARRISON LEVA, JUAN TREJO, TAE JIN KIM, CARLOS HIDROVO, University of Texas at Austin — Poly(N-isopropylacrylamide) has become ubiquitously known as a "smart" polymer, showing many promising applications in tissue engineering and drug delivery systems. These applications are particularly reliant on its trenchant, thermally induced hydrophilic-hydrophobic transition that occurs at the lower critical solution temperature (LCST). This feature imparts the pNIPAM programmable adsorption and release capabilities, thus eliminating the need for additional enzymes when removing cells from pNIPAM coated surfaces and leaving the extracellular matrix proteins of the cells largely untouched. The dependence of the LCST on molecular weight, solvent systems, and various salts has been studied extensively. However, what has not been explored is the effect of humidity on the characteristic properties of the polymer, specifically the LCST and the magnitude of the hydrophilic-hydrophobic transition. We studied the surface energy variation of pNIPAM as a function of humidity by altering the absolute humidity and keeping the ambient temperature constant. Our experiments were conducted inside a cuboidal environmental chamber with control over the temperature and humidity inside the chamber. A controlled needle was employed to dispense size-regulated droplets. Throughout this process, a CCD camera was used to image the droplet and the static contact angle was determined using image processing techniques. The behavior of pNIPAM as a function of humidity is presented and discussed.

11:09AM H7.00004 Particle Image Velocimetry During Injection Molding , THOMAS BRESS[1], Exponent Failure Analysis Associates, DAVID DOWLING, Dept. of Mechanical Engineering, University of Michigan — Injection molding involves the unsteady non-isothermal flow of a non-Newtonian polymer melt. An optical-access mold has been used to perform particle image velocimetry (PIV) on molten polystyrene during injection molding. Velocimetry data of the mold-filling flow will be presented. Statistical assessments of the velocimetry data and scaled residuals of the continuity equation suggest that PIV can be conducted in molten plastics with an uncertainty of ± 2 percent. Simulations are often used to model polymer flow during injection molding to design molds and select processing parameters but it is difficult to determine the accuracy of these simulations due to a lack of in-mold velocimetry and melt-front progression data. Moldflow was used to simulate the filling of the optical-access mold, and these simulated results are compared to the appropriately-averaged time-varying velocity field measurements. Simulated results for melt-front progression are also compared with the experimentally observed flow fronts. The ratio of the experimentally measured average velocity magnitudes to the simulation magnitudes was found on average to be 0.99 with a standard deviation of 0.25, and the difference in velocity orientations was found to be 0.9 degree with a standard deviation of 3.2 degrees.

[1]formerly at the University of Michigan

11:22AM H7.00005 Quantitative analysis of the debonding structure of soft adhesives[1] , MATTEO NICOLI, Lab. PMC - Ecole Polytechnique, FRANCOIS TANGUY, COSTANTINO CRETON, Lab. PPMD - ESPCI — Pressure sensitive adhesives (PSAs) are viscoelastic or viscoplastic materials that adhere to a substrate upon the application of light pressure. The debonding mechanism is an interfacial process due to different phenomena, e.g. the creation of cavities and fibrils, the propagation of interfacial cracks and the lateral invasion of air fingers. The studies of adhesive performances of PSAs are carried out through the probe tack tester. We developed a boundary recognition algorithm to analyze the top-view images from probe tack experiments, allowing us to detect the nucleation of cavities, track their growth and measure various geometrical quantities. We tested three PSAs with different viscoelastic features, ranging from a more liquid to a more elastic behavior, at two debonding velocities ($1, 10\,\mu m s^{-1}$). We measured the load bearing area and estimated the magnitude of the shear stress from the nominal force and the uniaxial tensile stress. From the characterization of the projected radius of each bubble, we tested the assumption of spherical growth of these cavities after the onset of their nucleation. The probe tack test combined with our methodology provides valuable data to understand the interfacial processes leading to the debonding of PSAs.

[1]We acknowledge funding from an FP7 EU collaborative project (MODIFY- 228320).

11:35AM H7.00006 Shear Banding in Polymer Solutions with a Monotonic Constitutive Curve , MICHAEL CROMER, MICHAEL VILLET, GLENN FREDRICKSON, GARY LEAL, University of California, Santa Barbara — Shear banding is a well-documented phenomenon occurring in the flow of various complex fluids, e.g. wormlike micellar solutions. In recent years, experiments have revealed that shear banding can occur in entangled polymer solutions. To model this behavior it has been assumed, for over 40 years, that an underlying non-monotonic constitutive curve is required to theoretically generate steady shear banding. It is widely believed, however, that the underlying constitutive curve for polymer solutions is strictly increasing. In this we talk we show that the linear instability of a monotonic curve driven by the Helfand-Fredrickson mechanism of polymer moving up stress gradients results in a shear-banded profile with a non-uniform concentration distribution. In addition, a subcritical, nonlinear instability exists resulting in multiple steady states, which agrees with a recent experimental observation.

11:48AM H7.00007 Rheological hysteresis in soft glassy materials , THIBAUT DIVOUX, VINCENT GRENARD, SEBASTIEN MANNEVILLE, Ecole Normale Superieure de Lyon — The nonlinear rheology of a soft glassy material is captured by its constitutive relation, shear stress vs shear rate, which is most generally obtained by sweeping up or down the shear rate over a finite temporal window. For a huge amount of complex fluids, the up and down sweeps do not superimpose and define a rheological hysteresis loop. By means of extensive rheometry coupled to time-resolved velocimetry, we unravel the local scenario involved in rheological hysteresis for various types of well-studied soft materials. Building upon a systematic experimental protocol, we introduce two observables that quantify the hysteresis in macroscopic rheology and local velocimetry respectively, as a function of the sweep rate δt^{-1}. Strikingly, both observables present a robust maximum with δt, which defines a single material-dependent timescale that grows continuously from vanishingly small values in simple yield stress fluids to large values for strongly time-dependent materials. In line with recent theoretical arguments, these experimental results hint at a universal timescale-based framework for soft glassy materials, where inhomogeneous flows characterized by shear bands and/or wall slip play a central role.

12:01PM H7.00008 Structure evolution in electrorheological fluids[1] , BIAN QIAN, AHMED HELAL, MARIA TELLERIA, Massachusetts Institute of Technology, MIKE MURPHY, MARC STRAUSS, Boston Dynamics, GARETH MCKINLEY, ANETTE HOSOI, Massachusetts Institute of Technology — Enhanced knowledge of the transient behavior and characteristics of electrorheological (ER) fluids subject to time dependent electric fields carries the potential to advance the design of fast actuated hydraulic devices. In this study, the dynamic response of electrorheological fluid flows in rectilinear microchannels was investigated experimentally. Using high-speed microscopic imaging, the evolution of particle aggregates in ER fluids subjected to temporally stepwise electric fields was visualized. Nonuniform growth of the particle structures in the channel was observed and correlated to field strength and flow rate. Two competing time scales for structure growth were identified. Guided by experimental observations, we develop a phenomenological model to quantitatively describe and predict the evolution of microscale structures and the concomitant induced pressure gradient.

[1]This work is supported by DARPA M3.

12:14PM H7.00009 Experimental Study of Settling of Spherical Particles in Unbounded and Confined Shear Thinning Viscoelastic Fluids , MUKUL M. SHARMA, SAHIL MALHOTRA, The University of Texas at Austin — An experimental study is performed to understand and quantify settling velocity of spherical particles in unbounded and confined surfactant-based shear thinning viscoelastic fluids. Experimental data is presented to show that elastic effects can increase or decrease the settling velocity of particles, even in the creeping flow regime. A significant drag reduction occurs with increase in Weissenberg number. This is followed by a transition to increasing drag at higher Weissenberg numbers. A new correlation is presented for the sphere settling velocity in unbounded viscoelastic fluids as a function of the fluid rheology and the proppant properties. The wall factors for sphere settling velocities in viscoelastic fluids confined between solid parallel plates are calculated from experimental measurements made over a range of Weissenberg numbers. Results indicate that elasticity reduces the effect of the confining walls and this reduction is more pronounced at higher ratios of the particle diameter to spacing between the walls. Shear thinning behavior of fluids is observed to reduce the retardation effect of the confining walls. A new empirical correlation for wall factors for spheres settling in a viscoelastic fluid confined between two parallel walls is presented.

12:27PM H7.00010 Micro-Macro Simulation of Viscoelastic Fluids in Three Dimensions , ALEXANDER RÜTTGERS, MICHAEL GRIEBEL, Institute for Numerical Simulation, University of Bonn — The development of the chemical industry resulted in various complex fluids that cannot be correctly described by classical fluid mechanics. For instance, this includes paint, engine oils with polymeric additives and toothpaste. We currently perform multiscale viscoelastic flow simulations for which we have coupled our three-dimensional Navier-Stokes solver NaSt3dGPF with the stochastic Brownian configuration field method on the micro-scale. In this method, we represent a viscoelastic fluid as a dumbbell system immersed in a three-dimensional Newtonian liquid which leads to a six-dimensional problem in space. The approach requires large computational resources and therefore depends on an efficient parallelisation strategy. Our flow solver is parallelised with a domain decomposition approach using MPI. It shows excellent scale-up results for up to 128 processors. In this talk, we present simulation results for viscoelastic fluids in square-square contractions due to their relevance for many engineering applications such as extrusion. Another aspect of the talk is the parallel implementation in NaSt3dGPF and the parallel scale-up and speed-up behaviour.

Monday, November 19, 2012 10:30AM - 12:27PM –

Session H8 Drops VII 25A - Chair: Sushanta Mitra, University of Alberta

10:30AM H8.00001 Coffee Stain Effect with Liquid Droplets[1] , SUSHANTA MITRA, SIDDHARTHA DAS, University of Alberta — We discuss the dynamics of immiscible bidispersed oil droplets that are suspended in an evaporating water sessile drop. Therefore, in contrast to classical coffee stain problem, the depositing "particles" are replaced by microscopic oil droplets – hence, we discuss a liquid-droplet coffee stain phenomenon. We show experimentally that unlike colloidal particles in a classical coffee stain problem, liquid oil droplets cannot reach the three phase contact line (TPCL) due to the aversion of the oil droplets to form finite oil-air interface in water medium. Therefore, the oil droplets get positioned at a finite distance from the TPCL. We call this distance the "enclosure" distance, which being a function of the droplet size, triggers a spontaneous size-based oil droplet separation. In addition, the "enclosure" effect is a function of the surface energies of the oil droplet and the rate of evaporation. We develop a theory to describe this effect, and the results show excellent agreement with the experimental findings.

[1]NSERC Banting Postdoctoral Fellowship for S. Das

10:43AM H8.00002 The dynamics of drops coating the underside of a flexible wall[1] , CRASTER RICHARD, ALEX WRAY, DEMETRIOS PAPAGEORGIOU, OMAR MATAR, Imperial College London — Lister et al., 2009, showed that a thin fluid coating the underside of a ceiling (a model which extends in particular the works of Hammond, 1983, and Lister et al., 2005) can give rise to pendent drops. If these are fixed in place by boundary conditions, they drain to give drops of constant pressure surrounded by annular trenches. These authors also showed that, on larger domains starting from an initial perturbation, these drops will undergo a self-induced quasi-steady translation. This is driven by the release of gravitational potential energy as the fluid in the film falls into the drop. The speed and growth of these drops is accessible to analytical computation by the self-similar study of the thin trenches surrounding them, and matching to far external conditions. The subsequent dynamics are intricate, allowing for coalescence (not seen in 1 dimension) as well as complex drop-drop interactions. We extend this model to allow for the ceiling to be a flexible substrate, and also to account for inertial effects in the drops. We then investigate the effect this has on the dynamics of the drops.

[1]EPSRC DTA

10:56AM H8.00003 Slip or not slip? A comparison of models applied to contact line motion , NIKOS SAWA, DAVID SIBLEY, SERAFIM KALLIADASIS, Department of Chemical Engineering, Imperial College London — The motion of a contact line is examined through the spreading of a thin two-dimensional droplet on a planar substrate to compare the contemporary model based on interface formation of Shikhmurzaev to more prevalent models in the literature. The implementation of the long-wave approximation for the droplet thickness in a quasistatic spreading regime affords an insight to the interface formation model behaviour, principally that the evolution of the droplet radius reduces to an equivalent expression for a slip model when the prescribed dynamic contact angle has a velocity dependent correction to its static value. This result is found both in the original interface formation model formulation and for a more recent version, where mass transfer from bulk to surface layers is accounted for through the boundary conditions. Various features, such as the pressure behaviour and rolling motion at the contact line, are critically analysed.

11:09AM H8.00004 The Effect of Droplet Inertia on Combined Gravitational and Thermocapillary Interactions of Contaminated Spherical Drops , MICHAEL ROTHER, University of Minnesota Duluth — The motion of two spherical drops under the influence of gravity and a vertical temperature gradient in the presence of bulk insoluble, nonionic surfactant is considered in the limit of an incompressible surfactant film. In this limit the surfactant coverage is nearly uniform. Thermal convection is negligible, and flow conditions are such that droplet inertia is important, while inertia due to the surrounding matrix fluid can be neglected. In terms of dimensionless parameters, these conditions imply that the Stokes number may be order unity or larger, while the Reynolds number remains small. Physically, such conditions generally mean that the density of the surrounding fluid is much less than the drop density and that the drop radii are typically in the range of 5 to 50 micrometers. In the case where both the Stokes and Reynolds numbers are small, closed and retrograde trajectories are observed. The effect of droplet inertia is to destroy symmetry in the trajectories and reduce the collision-forbidden region of the parameter space. Van der Waals forces are also taken into account. A practical application of this research is in the study of raindrop growth, where the collision efficiencies fall between those for clean drops and solid spheres.

11:22AM H8.00005 ABSTRACT WITHDRAWN –

11:35AM H8.00006 Effects of surface wettability and edge geometry on drop motion through an orifice[1] , ANKUR BORDOLOI, ELLEN LONGMIRE, Aerospace Engineering and Mechanics, University of Minnesota — In geothermal energy recovery and CO_2 sequestration, drops move through a porous structure by displacing a surrounding liquid. Both the pore geometry and surface wettability influence the drop motion. We simplify the pore structure to a thin plate with a circular orifice. The plate is held horizontally inside a rectangular tank filled with silicone oil. Drops of water/glycerin with Bond numbers (Bo) of 1-10 are released above and axisymmetric to the orifice, encountering the plate after reaching their terminal speed. We use high speed imaging to examine the effects of orifice-to-drop diameter ratio (d/D), orifice surface wettability (hydrophilic/hydrophobic) and edge geometry on the passage of drop fluid through the orifice. We generate regime maps for d/D and Bo delineating domains of drop capture, passage, and passage with breakup. For $d/D < 1$, sharp edges are observed to yield contact between the drop and orifice so that surface wettability influences the subsequent dynamics. On the other hand, rounded edges appear to prevent direct contact so that the dynamics are unaffected by the surface wettability.

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

11:48AM H8.00007 A theory of wall-induced lateral migration of a drop in shear: effects of drop inclination and viscoelasticity[1] , KAUSIK SARKAR, George Washington University — Recently, migration of suspended particles, drops, polymers and biological cells have assumed importance in microfluidic separation and filtration assays. A rigid sphere in shear does not move cross-stream due to the reversibility in a Stokes flow. Deformation, as well as inertia and viscoelasticity, breaks the reversibility and leads to lateral migration away from a nearby wall. There have been algebraically complex perturbative analyses of the moving boundary problem [e.g. Chan and Leal 1979, JFM 92,131] to determine the migration velocity. However, the underlying physics remains unclear. Here, we show that the migration is induced by the image stresslet field, as was also indicated earlier by Smart and Leighton [1991, Phys. Fluid A, 3, 21]. We relate the stresslet field to the Interface tensor, and investigate the effects of drop inclination. In contrast to a plausible notion asserted also in the literature, that reduced inclination (increased alignment with flow) decreases migration, it is shown here that reduced inclination increases the stresslet and thereby the migration velocity. A semi-analytical expression of migration velocity based on numerically computed stresslet will be compared against simulation and the results will be discussed.

[1]Partially supported by NSF

12:01PM H8.00008 Computational investigation of spreading and arrest of molten liquid on a solid substrate , REZA BAGHAEI LAKEH, FARYAR TAVAKOLI, PIROUZ KAVEHPOUR, University of California at Los Angeles — In this study, the non-isothermal spreading and solidification of a molten liquid droplet on a cold solid substrate have been explored. A computational effort was undertaken to investigate the dynamic and thermal characteristics of liquid spreading and subsequent arrest of the liquid droplet on a cold surface. The phenomena of spreading of the liquid droplet, subsequent solidification, and arrest of the droplet are not well understood. The developed model simulates the spreading and arrest phenomena by computational fluid dynamic techniques. A parametric study of the arrest contact angle, and base diameter were studied computationally. Our results show that the arrest contact angle and base radius are function of the Stefan number and Prandtl number of melt.

12:14PM H8.00009 Physics of Spreading and Arrest of Molten Liquid on Solid Substrates , FARYAR TAVAKOLI, University of California, Los Angeles, STEVEN DAVIS, Northwestern University, PIROUZ KAVEHPOUR, University of California, Los Angeles — The physics of non-isothermal spreading followed by phase change, unlike universal equations established for isothermal spreading, is still a mystery. A plethora of applications such as coating technology, rapid prototyping, 3D printing and plastic electronics involve molten droplets spreading on cold substrate surfaces. Better control of these processes requires fundamental understanding of heat transfer and fluid flow that transpire during the spreading and solidification of liquid droplets. The present work focuses on the dynamic and thermal characteristics of liquid spreading and subsequent arrest. Spreading of liquid was recorded and evolution of liquid spread diameter and liquid-solid contact angle were measured from the recordings of a high-speed digital camera. After solidification initiation at the basal plane, a liquid drop is pinned to a solid substrate showing fixed footprint and contact angle. Arrested contact angle (θ*) and arrested base diameter (D*) are evaluated against two main contributing variables: fluid flow rate and Stefan number. We have developed a power law that indicates the arrested contact angle and base diameter are not single-valued for given substrate temperature, but are strongly dependent on flow rate and the surface characteristics.

Monday, November 19, 2012 10:30AM - 12:40PM –

Session H9 Interfacial/Thin Film Instability IV 25B - Chair: Jacopo Seiwert, Institut de Physique de Rennes

10:30AM H9.00001 Influence of Heat Transfer on Stability of Newtonian and Non-Newtonian Extending Films , ZHEMING ZHENG, OLUS BORATAV, CHUNFENG ZHOU, Corning Incorporated, CORNING INCORPORATED TEAM — We consider the stability of Newtonian and Non-Newtonian extending films under local or global heating or cooling conditions. We derive the thickness-averaged mass, momentum and energy equations (energy equation including both convective and radiative heat transfer) and analyze the stability using eigenvalue analysis. We show that the influence of heating and cooling on stability strongly depends on the magnitude of the Peclet number. Examples of stabilization or destabilization of heating or cooling are shown for very small and very large Peclet numbers. The impact of three non-Newtonian (viscoelastic) models (Maxwell, PTT and Giesekus) on stability results is also discussed.

10:43AM H9.00002 Fluid pumping in thin films using thermal waves , ALEXANDER ALEXEEV, Georgia Institute of Technology, Atlanta, GA, 30332, USA, WENBIN MAO, Georgia Institute of Technology, ALEXANDER ORON, Technion-Israel Institute of Technology, Haifa, 32000, Israel — Open microfluidic devices are used in many applications, including bio-sensing, molecule manipulation, and microchip cooling. We use direct numerical simulations of the full continuity, Navier-Stokes, and energy equations along with the analysis based on the long-wave theory to examine the dynamics of thin films on substrates with periodic heating that propagates in form of thermal waves along the substrate. Using these two modeling techniques, we probe how the periodic thermal wave can be harnessed to induce and regulate directed fluid flows along the substrate. Furthermore, we study the stability of these solutions to identify the parameter range providing robust fluid pumping in open microfluidic systems.

10:56AM H9.00003 Two-phase investigation of hydrothermal waves in saturated atmospheres[1] , KHELLIL SEFIANE, PEDRO SAENZ, PRASHANT VALLURI, University of Edinburgh, GEORGE KARAPETSAS, University of Thessaly, OMAR MATAR, Imperial College London — A liquid layer subject to a sufficiently large thermal gradient along its interface is prone to depart from its equilibrium state and to develop into an oscillatory regime whose features may differ notably from the original state. In shallow liquid pools, the preferred instability mode is obliquely-travelling hydrothermal waves (HTWs). We investigate this Marangoni-driven flow by means of two-phase direct numerical simulations in 3D with the interface captured via the volume-of-fluid method. Validated against experiments (Riley et al. 1998) and linear theory (Smith & Davis 1983), the results reveal the highly-intricate spatio-temporal evolution of the instabilities and the presence of interfacial waves tightly coupled with the HTWs. The instability's development and the interdependencies amongst HTWs, interface deformations and bulk flows (liquid and gas phases) are thoroughly investigated for the linear (early times) and non-linear (late times) stages. We also elucidate the heat-transfer mechanism across the interface which is significantly affected by the propagating disturbances.

[1]EPSRC DTA

11:09AM H9.00004 Thermocapillarity driven Instabilities in thin liquid layers subject to long-wave analysis , ANEET NARENDRANATH, Michigan Technological University, JAMES HERMANSON, University of Washington, Seattle, ALLAN STRUTHERS, ROBERT KOLKKA, JEFFREY ALLEN, Michigan Technological University — An evolution equation describing the dynamics of an evaporating liquid film has previously been developed from the governing equations of fluid dynamics after the application of the lubrication approximation and the choice of a viscous time scale. The authors have solved the evaporating liquid film evolution equation with a validated numeric program. The role of domain size and thermocapillarity on the formation of secondary finger like structures is studied. The effect that gravity has on the formation of these finger patterns is evaluated. It is observed that the formation of secondary structures is strongly tied to a balance between destabilizing thermocapillarity and stabilizing surface tension. The secondary structures are amplified in a zero gravity environment.

11:22AM H9.00005 Instability of a fluctuating membrane driven by an AC electric field , JACOPO SEIWERT, Institut de Physique de Rennes, PETIA VLAHOVSKA, Brown University — We consider theoretically the stability of a biomimetic planar membrane stressed by a perpendicularly applied time-varying electric field. The membrane, which separates two fluids, is modeled as a leaky capacitor. We investigate the influence of membrane electrical properties, as well as those of the surrounding fluids, on membrane stability. Our linear stability analysis shows that unlike what happens with DC electric fields, a purely capacitive membrane can be destabilized in an AC electric field. The theory highlights that the instability originates from electric pressure exerted on the membrane.

11:35AM H9.00006 Electrostatically induced long-wave dynamics in moderately conducting annular flows[1] , ALEX WRAY, DEMETRIOS PAPAGEORGIOU, OMAR MATAR, Imperial College London — Annular flows on both the inner and outer walls of cylinders have a significant number of practical applications; these range from printing to fluid stabilisation, to the augmentation of heat and mass transfer. It is important to understand the spatiotemporal dynamics of the interface in order to facilitate the possible control of the flow and efficiency of underlying processes. We investigate the evolution and stability of a viscous fluid layer wetting the surface of a cylinder and surrounded by a gas. The inner cylinder is an electrode kept at constant voltage. A second concentric electrode, whose potential is allowed to vary as a function of both space and time, encloses the system. This induces electrostatic forces at the interface in competition with surface tension and viscous stresses. Asymptotic methods are use to derive a long-wave system of evolution equations, valid for moderate conductivities. The resulting system is investigated both analytically and numerically via a systematic parametric study.

[1]EPSRC DTA

11:48AM H9.00007 Control of complex dynamics in highly conducting thin annular films[1] , DEMETRIOS PAPAGEORGIOU, ALEX WRAY, OMAR MATAR, Imperial College London — Thin, highly conducting annular films represent a canonical scenario giving rise to both mathematical interest and practical physical relevance. A precise and complete understanding of the dynamics of such a flow is vital for its effective manipulation. We look at the case of a thin, highly conductive film on the outer surface of a conductive cylinder, with a potential difference set up between the coating cylinder and second, concentric cylinder. The disparity of electrical material properties in the two regions induces additional electric stresses at the interface. Asymptotic methods are used to derive a thin film type equation governing this situation. These lead to an additional term in the gravitationally modified Hammond equation, with a modified mobility coefficient, which can either augment disturbances, or drive them to cessation. We discuss the intricate effect that this has on both the linear and nonlinear regimes in one dimension. We also discuss briefly the full two-dimensional dynamics via transient numerical simulations.

[1]EPSRC DTA

12:01PM H9.00008 Phase separation patterns in irradiated thin liquid films due to optical interference effects , FUMIHIRO SAEKI, SHIGEHISA FUKUI, HIROSHIGE MATSUOKA, Tottori University — The pattern formation in irradiated thin liquid films on solid substrates is investigated within the framework of the long-wave approximation. The focus is placed on a transparent film/absorbable substrate system irradiated by a monochromatic wave with laterally uniform intensity distribution. The evolution of the film surface profile is described by an equation of Cahn-Hilliard type, and the free energy density that is a function of the film thickness has local minima due to optical interference between waves reflected from the gas-liquid and liquid-solid interfaces. Therefore, a small perturbation develops into a phase separation pattern if the unperturbed uniform state is unstable.

12:14PM H9.00009 The Nanoworld Beyond Bénard Instability: Comparison Between Theory and Experiment , KEVIN FIEDLER, SANDRA TROIAN, California Institute of Technology, MC 128-95, Pasadena, CA 91125 — The spontaneous growth of fluid elongations in a viscous nanofilm whose free surface is held in close proximity to a cooler substrate has been attributed to three different mechanisms. Linear stability analyses in the long wavelength approximation indicate that such formations arise either from fluctuations in the electrostatic attraction between the fluid interface and the opposing substrate, the acoustic phonon pressure acting on the film, or thermocapillary stresses along the free surface. The latter mechanism represents the long wavelength limit of the Bénard-Marangoni problem in the absence of a critical number. Model validation requires direct comparison between experiment and linear stability theory, which necessitates that the film structuring process be analyzed at very early times. Previous measurements in our group of the fastest growing wavelength exceeded theoretical predictions of the thermocapillary model by a factor of 2 to 3. More detailed studies of the formation process highlight the importance of obtaining measurements at very early times. Recent data obtained by varying the substrate separation distance, initial film thickness and overall temperature difference indicate much improved agreement with the thermocapillary model.

12:27PM H9.00010 Thermocapillary motion of a droplet on an inclined plate[1] , GEORGE KARAPETSAS, University of Thessaly, KIRTI SAHU, IIT Hyderabad, OMAR MATAR, Imperial College London — We examine the dynamics of a droplet spreading on an inclined solid surface in the presence of constant wall thermal gradients. We use lubrication theory in combination with the Karman-Polhausen integral method to simplify the governing equations for the droplet motion and energy conservation leading to coupled evolution equations for the drop thickness and average temperature. An important feature of the spreading model developed here is the behaviour of the drop at the contact line; this is modeled using a constitutive relation, which is dependent on the local temperature of the wall. We use a finite-element formulation to obtain numerical solutions of the evolution equations and carry out a full parametric study. We investigate the various types of behaviour encountered due to the interplay of Marangoni stresses, gravity and the dynamics of the contact line.

[1]EPSRC Grant number EP/E046029/1

Monday, November 19, 2012 10:30AM - 12:14PM –

Session H10 Instability: Jets, Wakes and Shear Layers V 25C - Chair: Tobias Schneider, Max Planck Institute for Dynamics and Self-Organization

10:30AM H10.00001 Local and global states in a reduced model for shear flows , MATTHEW CHANTRY, RICH KERSWELL, University of Bristol — A large body of work in shear flow turbulence has involved finding fixed point, travelling wave or periodic orbits, whose manifolds shape the phase space of turbulence. Most of the solutions found fill their periodic domains and are therefore not helpful in understanding localized turbulence. Recent work has pointed to connections between these spatially periodic solutions and versions of these solutions localized in at least one spatial dimension. In this work we consider a nine mode PDE model for shear flow turbulence, which depends on one spatial direction, and attempt to understand this connection between localized and periodic solutions.

10:43AM H10.00002 Sensitivity analysis of the periodic orbits of plane Couette flow , ONOFRIO SEMERARO, Linne' Flow Centre, KTH Mechanics - 10044, Stockholm - Sweden, FLAVIO GIANNETTI, DIMEC, Universita' di Salerno - 84084, Fisciano (SA) - Italy, LUCA BRANDT, Linne' Flow Centre, KTH Mechanics - 10044, Stockholm - Sweden — In the last decade, concepts from dynamical systems have been applied to gain insight onto transition to turbulence in shear flows, such as pipe and plane Couette flow. For the latter, several nontrivial equilibrium solutions, travelling waves and periodic orbits were found, enabling the analysis of transition from a new perspective. In this contribution, we study the sensitivity of unstable periodic orbits in Couette flow. These limit cycles are dominated by the self-sustained interaction between vortices and streaks. By applying the Floquet analysis, we calculate the flow Lyapunov exponent; investigating the most unstable Floquet modes and the related adjoint modes, we analyze the eigenvalue sensitivity to localized structural perturbations over the orbit period. Preliminary results confirm that the core of the instability coincides with the region where the streaks are bent, in agreement with previous numerical results. In the final contribution the analysis will be completed by considering the sensitivity of the limit-cycle frequency and amplitude to feedback forcing.

10:56AM H10.00003 The edge in models of shear flows , NORMAN LEBOVITZ, University of Chicago, GIULIO MARIOTTI, Boston University — A characteristic feature of the onset of turbulence in shear flows is the appearance of an "edge," a codimension-one invariant manifold that separates orbits that decay rapidly to the laminar state from orbits that decay more slowly. We investigate its structure by considering a series of models of successively higher dimension. We hope in this way to isolate geometric features that are robust under the increase of dimension and are therefore candidates for extrapolation to arbitrarily high dimension. We find in the cases considered that there are extensive ranges of the Reynolds number in which all or part of the boundary of the basin of attraction of the laminar state indeed has the character of an edge. The edge is also the stable manifold of an edge state (a "lower-branch" state). An important feature of the edge is that it lies in a region of phase space which, while unbounded in some directions, is bounded in others. This allows orbits on either side of it to connect to the laminar state. The boundedness of the edge is associated with the presence of a further invariant ("upper-branch") state.

11:09AM H10.00004 Infinity-norm optimal perturbations in 2D plane Poiseuille flow , DIMITRY P.G. FOURES, COLM-CILLE P. CAULFIELD, DAMTP - University of Cambridge, PETER J. SCHMID, LadHyX - Ecole Polytechnique — Since the emergence of the concept of non-modal stability analysis in the early 90's, many efforts have been made in order to identify the optimal linear mechanisms at stake in the finite-time triggering of highly energetic perturbations in a linearly stable flow. The objective functional typically involved an integrated measure of the total perturbation KE over the domain. In some circumstances however, one may be interested in identifying perturbations which lead to a maximum localized peak value of KE. This problem requires a departure from maximization of the usual quadratic norm (an inherently global measure) of the velocity field. As a demonstration case, we choose to investigate ∞-norm optimal perturbations in a 2D plane Poiseuille flow at $Re = 4000$. We show that for any optimization time horizon, two branches of solutions exist, either associated with perturbations localized in the centre of the domain ("centre mode") or close to the domain boundaries ("wall mode"). We find that for $T \lesssim 0.5T_O$ and $T \gtrsim 2T_O$ (with T_O the global optimal time for total KE), the wall modes are more efficient at producing highly energetic localized perturbations, while centre modes are optimal for intermediate times $0.5T_O \lesssim T \lesssim 2T_O$ only.

11:22AM H10.00005 Localization in shear flow turbulence , TOBIAS M. SCHNEIDER, Max Planck Institute for Dynamics and Self-Organization, JOHN F. GIBSON, University of New Hampshire, JOHN BURKE, Boston University — Transitional turbulence in shear flows such as Couette flow is often characterized by spatio-temporal patterns and the coexistence of laminar and turbulent flow. Some of those spatial features are captured by new classes of spatially localized exact coherent structures. They are related to their known periodic counterparts and some show bifurcation structures very similar to those observed in simpler pattern-forming systems. Characterizing those solutions and generalizing the dynamical systems view of turbulence to capture the full spatio-temporal dynamics is a step towards developing a general theory of patterns in shear flows.

11:35AM H10.00006 New exact coherent states in plane Poiseuille flow , MASATO NAGATA, KENGO DEGUCHI, Kyoto University — Two new classes of traveling wave solution are found in plane Poiseuille flow by continuing the stationary and traveling hairpin vortex states in plane Couette flow. One of them, referred to as MS hereafter, arises from a saddle-node bifurcation, characterized by two planes of mirror-symmetry perpendicular to the span-wise direction. The second new class solution, referred to as AS hereafter, bifurcates by breaking the mid-plane symmetry of the first class. Both MS and AS are characterized by two quasi-stream-wise low-speed streaks within one span-wise period. The low-speed streaks are aligned with the vertical planes of mirror symmetry, with their width varying in a varicose fashion in the stream-wise direction. These streaks appear close to both top and bottom channel walls for MS, and to only one of the channel walls for AS. We find that the Reynolds numbers at the saddle-node bifurcation for MS and AS are smaller than that of the exact coherent state in plane Poiseuille flow known to date found by Waleffe (2003).

11:48AM H10.00007 Phase transition to sustained turbulence in pipe flow , MUKUND VASUDEVAN, MARCO VASSALLO, BJORN HOF, Max Planck Institute for Dynamics and Self-Organization — Turbulence in pipe flow can first arise at Reynolds numbers somewhat below 2000. Here turbulent structures ("puffs") are localized and have a finite lifetime. Turbulence can also proliferate through puff splitting and it has recently been proposed that turbulence overall becomes sustained when this spreading process outweighs the decay of individual structures. In the present study we measure the decay rate of turbulent puffs in two different set ups: In the first the pressure difference across the experiment is held fixed. In the second the flow is driven a piston system that enforces a constant flow rate. Measurements in both set ups are in excellent qualitative agreement and confirm that individual turbulent puffs are intrinsically transient and decay following a memoryless process. Exploiting the memoryless nature a pipe with quasi periodic boundary conditions is constructed allowing indefinitely long observation times of puff sequences. This method for the first time allows to directly measure the asymptotic evolution of the turbulent flow and to determine the equilibrium turbulent fraction close to the critical point.

12:01PM H10.00008 Evolution of K- and H-type structures in a spatially evolving channel flow[1] , ALEC KUCALA, SEDAT BIRINGEN, SCOTT WAGGY, Department of Aerospace Engineering Sciences; University of Colorado; Boulder, CO 80309 — A fully parallel, direct numerical simulation (DNS) is performed on the full time-dependent, three-dimensional Navier-Stokes equations in a spatially developing plane-channel flow at $Re = 10,000$, in which two-dimensional eigenfunctions based on the solution of the Orr-Sommerfeld equation are introduced at the inlet with uniform random noise $A_r < 10^{-3}$ added along the spanwise and wall-normal directions. The flow is allowed to "choose" a path to secondary instability, K-type (after Klebanoff) or H-type (after Herbert), depending on the amplitude of the 2D disturbance. Detailed analysis of the spatial evolution of the primary, fundamental and subharmonic modes are presented to examine the path of secondary transition. Flow visualizations using Lagrangian coherent structures (LCS) are shown, giving physical insight into the coherent structures involved in the breakdown of laminar flow in a plane-channel.

[1]NSF Award #1131802

Monday, November 19, 2012 10:30AM - 12:27PM –

Session H11 Bubbles III 26A - Chair: Marc Smith, Georgia Institute of Technology

10:30AM H11.00001 Bubble contraction in free-boundary Hele–Shaw flow with surface tension and kinetic undercooling regularisation , MICHAEL DALLASTON, SCOTT MCCUE, Queensland University of Technology — When an inviscid bubble expands into a viscous fluid in a Hele-Shaw cell, the bubble boundary is unstable, in general forming long fingers (the Saffman–Taylor instability). In order to make the problem well-posed, a regularising boundary effect must be included. The most widely studied of these are surface tension, which penalises high curvatures, and kinetic undercooling, which penalises high velocities. Both these effects act as a stabilising influence on the free boundary. Less attention has been paid to the case of contracting bubbles, which shrink to a single point (or points) in finite time. In this case, the two effects are in competition, as surface tension stabilises the boundary, while kinetic undercooling destabilises it. This leads to bifurcation behaviour in the asymptotic (near-extinction) shape of the bubble as the relative strengths of the two effects are varied. In particular, there is a critical range of parameter values for which both circular and slit-type bubbles are stable, with a third (unstable) oval-type shape also present. We discuss some numerical and analytic techniques for solving the full free boundary problem and for exploring this interesting extinction behaviour.

10:43AM H11.00002 Cavitation propagation in water under tension[1] , XAVIER NOBLIN, YANN YIP CHEUNG SANG, MATHIEU PELLEGRIN, LPMC, CNRS UMR 7336, Universite de Nice Sophia-Antipolis, Parc Valrose, 06108 Nice Cedex 2,, MATERIALS AND COMPLEX FLUIDS TEAM — Cavitation appears when pressure decreases below vapor pressure, generating vapor bubbles. It can be obtain in dynamical ways (acoustic, hydraulic) but also in quasi-static conditions. This later case is often observed in nature, in trees, or during the ejection of ferns spores [1]. We study the cavitation bubbles nucleation dynamics and its propagation in a confined microfabricated media. This later is an ordered array of microcavities made in hydrogel filled with water [2]. When the system is put into dry air, it dehydrates, water leaves the cavities and tension (negative pressure) builds in the cavities. This can be sustained up to a critical pressure (of order -20 MPa), then cavitation bubbles appear. We follow the dynamics using ultra high speed imaging. Events with several bubbles cavitating in a few microseconds could be observed along neighboring cells, showing a propagation phenomenon that we discuss.

[1] X. Noblin, N. O. Rojas, J. Westbrook, C. Llorens, M. Argentina, J. Dumais. The Fern Sporangium: A Unique Catapult. Science, 335, 1322, 2012.
[2] Tobias D. Wheeler and Abraham D. Stroock. The transpiration of water at negative pressures in a synthetic tree. Nature, 455, 208-212, 2008.

[1]ANR CAVISOFT 2010-JCJC-0407 01

10:56AM H11.00003 Vortex Cavitation on Delta-Wings , HARISH GANESH, STEVEN CECCIO, University of Michigan, Ann Arbor — Vortex cavitation experiments were carried out on delta wing with 70 degrees sweep back at various attack angles in a re-circulating water tunnel. The flow speed was maintained at a constant value of 4 m/s and the free stream pressure was varied to observe cavitation events ranging from inception to fully attached bubbles. By changing the free stream cavitation number two types of bubbles were observed; stationery bubbles that occupied an axial position for a sufficient time and travelling bubbles that convected with the flow. Variation of the vortex properties on a delta wing facilitates the occurrence of these two bubble types. Stationery bubbles, by the virtue of their equilibrium position and shape provide information about the axial and radial gradients of the flow respectively. A simple analysis based on equilibrium of a stationery bubble is used to illustrate the relation between the vortex properties and the stationery bubble properties. Acoustic signature of these bubbles were also measured and compared. Dye injection was also used to determine the breakdown location and it was found that vortex cavitation had no influence on the breakdown location. Finally, a relationship between stationery bubbles and the vortex properties is proposed based on the measurements and analysis.

11:09AM H11.00004 Enhancing cavitation with micromachined surfaces[1] , DAVID FERNANDEZ RIVAS, LAURA STRICKER, AALDERT G. ZIJLSTRA, HAN GARDENIERS, DETLEF LOHSE, University of Twente, ANDREA PROSPERETTI, Johns Hopkins University, MESOSCALE CHEMICAL SYSTEM GROUP COLLABORATION, PHYSICS OF FLUIDS GROUP COLLABORATION, DEPARTMENT OF MECHANICAL ENGINEERING COLLABORATION — When a silicon surface with micromachined pits submerged in a liquid is exposed to continuous ultrasound at 200 kHz, bubbles are ejected from the air filled cavities. Depending on the pressure amplitude different scenarios are observed, as the bubbles ejected from the micropits interact in complex ways with each other, and with the silicon surface. We have determined the size distribution of bubbles ejected from one, two and three pits for three different electrical power settings, and correlated them with sonochemical OH* radical production. Numerical simulations of the sonochemical conversion reaction rates were obtained using the empirical bubble size distributions and are compared with experimental results. Experimental evidence of shock wave emission from the microbubble clusters, deformed microbubble shapes, jetting and surface erosion are also presented.

[1]Financially supported through the project 07391 of the Technology Foundation STW, The Netherlands

11:22AM H11.00005 Instability arisen on condensing vapor bubble[1] , ICHIRO UENO[2], Dept. Mechanical Engineering, Fac. Science & Technology, Tokyo University of Science (TUS), RYOTA HOSOYA[3], Div. Mechanical Engineering, School of Science & Technology, Tokyo University of Science, CHUNGPYO HONG[4], Dept. Mechanical Engineering, Fac. Science & Technology, Tokyo University of Science — In the present study a special attention is paid to the growing and collapsing processes of vapor bubble injected into a subcooled pool; the authors try to extract the vapor-liquid interaction by employing a vapor generator that supplies vapor at designated flow rate to the subcooled pool instead of using a immersed heated surface to realize a vapor bubble by boiling phenomenon. This system enables ones to detect a spatio-temporal behavior of a single bubble of superheated vapor exposed to a subcooled liquid. The authors indicate the condensation rates as functions of the injection velocity of the vapor and the degree of subcooling of the pool. The authors indicate that an abrupt condensation of the injected vapor results in a fine disturbance over the vapor bubble surface before the collapse stage of the bubble. The wave number is sharply dependent on the degree of subcooling of the pool. The threshold of such a fine disturbance formation over the bubble corresponds with that the occurring condition of the maximum volume reduction rate of the vapor bubble.

[1]This work is supported by Grant-in-Aid for Scientific Research (B) (project #: 24360085) from Japan Society for the Promotion of Science (JSPS).
[2]also: Research Institute for Science & Technology (RIST), TUS
[3]present: Kawasaki Heavy Industries, Ltd.
[4]also: Research Institute for Science & Technology (RIST), TUS

11:35AM H11.00006 The behavior of vapor bubbles during boiling enhanced with acoustics and open microchannels[1] , THOMAS BOZIUK, MARC K. SMITH, ARI GLEZER, Georgia Institute of Technology — Boiling heat transfer on a submerged heated surface is enhanced by combining a grid of surface micromachined open channels and ultrasonic acoustic actuation to control the formation and evolution of vapor bubbles and to inhibit the instability that leads to film boiling at the critical heat flux (CHF). The microchannels provide nucleation sites for vapor bubble formation and enable the entrainment of bulk subcooled fluid to these sites for sustained evaporation. Acoustic actuation excites interfacial oscillations of the detached bubbles and leads to accelerated condensation in the bulk fluid, thereby limiting the formation of vapor columns that precede the CHF instability. The combined effects of microchannels and acoustic actuation are investigated experimentally with emphasis on bubble nucleation, growth, detachment, and condensation. It is shown that this hybrid approach leads to a significant increase in the critical heat flux, a reduction of the vapor mass above the surface, and the breakup of low-frequency vapor slug formation. A large-scale model of the microchannel grid reveals details of the flow near the nucleation site and shows that the presence of the microchannels decreases the surface superheat at a given heat flux.

[1]Supported by ONR.

11:48AM H11.00007 Development of a GPU and multi-CPU accelerated non-isothermal, multiphase, incompressible Navier-Stokes solver with phase-change[1] , CHRISTOPHER J. FORSTER, ARI GLEZER, MARC K. SMITH, Georgia Institute of Technology — Accurate 3D boiling simulations often use excessive computational resources – in many cases taking several weeks or months to solve. To alleviate this problem, a parallelized, multiphase fluid solver using a particle level-set (PLS) method was implemented. The PLS method offers increased accuracy in interface location tracking, the ability to capture sharp interfacial features with minimal numerical diffusion, and significantly improved mass conservation. The independent nature of the particles is amenable to parallelization using graphics processing unit (GPU) and multi-CPU implementations, since each particle can be updated simultaneously. The present work will explore the speedup provided by GPU and multi-CPU implementations and determine the effectiveness of PLS for accurately capturing sharp interfacial features. The numerical model will be validated by comparison to experimental data for vibration-induced droplet atomization. Further development will add the physics of boiling in the presence of acoustic fields. It is hoped that the resultant boiling simulations will be sufficiently improved to allow for optimization studies of various boiling configurations to be performed in a timely manner.

[1]Supported by ONR.

12:01PM H11.00008 Heat transfer enhancement in turbulent thermal convection close to the boiling point: Numerical simulations , R. LAKKARAJU, R.J.A.M. STEVENS, Physics of Fluids, University of Twente, The Netherlands, P. ORESTA, Department of Engineering for Innovation, University of Salento, Italy, F. TOSCHI, Department of Physics and Department of Mathematics and Computer Science, Eindhoven University of Technology, The Netherlands, C. SUN, Physics of Fluids, University of Twente, The Netherlands, R. VERZICCO, Department of Mechanical Engineering, University of Rome 'Tor Vergata', Italy, A. PROSPERETTI, Department of Mechanical Engineering, Johns Hopkins University, USA, D. LOHSE, Physics of Fluids, University of Twente, The Netherlands — We perform numerical simulation of turbulent Rayleigh-Bénard convection close to the boiling point of water, i.e., at 100° C and ambient pressure, in a cylinder of aspect ratio 1 for the Rayleigh number range $2 \times 10^6 \leq Ra \leq 5 \times 10^9$, modeling the vapor bubbles as two-way coupled point particles. We quantified the heat transfer enhancement as function of the number of bubbles, the degree of superheating (i.e., temperature excess of the plate as compared to the temperature of the bubble) and Ra. Heat transport is enhanced up to 6 times for low Ra and by to 2 times for high Ra through the presence of bubbles. Our results are consistent with the recent experimental findings of Zhong et al. (Phys. Rev. Lett. **102**, 124501, 2009), if one considers that the vapor bubble nucleation rate increases with the super heating.

12:14PM H11.00009 Numerical Investigation of Boiling , MICHAEL SAGAN, SEBASTIEN TANGUY, CATHERINE COLIN, IMFT — In this work, boiling is numerically investigated, using two phase flow direct numerical simulation based on a level set / Ghost Fluid method. Nucleate boiling implies both thermal issue and multiphase dynamics issues at different scales and at different stages of bubble growth. As a result, the different phenomena are investigated separately, considering their nature and the scale at which they occur. First, boiling of a static bubble immersed in an overheated liquid is analysed. Numerical simulations have been performed at different Jakob numbers in the case of strong density discontinuity through the interface. The results show a good agreement on bubble radius evolution between the theoretical evolution and numerical simulation. After the validation of the code for the Scriven test case, interaction of a bubble with a wall is studied. A numerical method taking into account contact angle is evaluated by comparing simulations of the spreading of a liquid droplet impacting on a plate, with experimental data. Then the heat transfer near the contact line is investigated, and simulations of nucleate boiling are performed considering different contact angles values. Finally, the relevance of including a model to take into account the evaporation of the micro layer is discussed.

Monday, November 19, 2012 10:30AM - 12:27PM –

Session H12 Vortex V 26B - Chair: Mark Stremler, Virginia Polytechnic Institute and State University

10:30AM H12.00001 Vortex evolution behind tandem cylinders under forced vibration , YINGCHEN YANG, University of Texas at Brownsville, TAYFUN AYDIN, ALIS EKMEKCI, University of Toronto — Flow past two circular cylinders in tandem arrangement has been studied experimentally employing the hydrogen bubble visualization technique. The two cylinders had the same diameter (D = 6.35 mm), and were subjected to forced in-phase vibration in the cross-flow direction. The Reynolds number based on the cylinder diameter was *Re* = 250. Both the vibration frequency (f_e) and center-to-center pitch ratio (P/D) were varied in certain ranges, whereas the vibration amplitude (A) was fixed at A/D = 0.25. The flow visualization resulted diverse and highly-repetitive vortex patterns. They were classified into two typical modes: a low-frequency mode and a high-frequency mode. The difference between the two modes is on the number of vortices formed per vibration cycle. For the low-frequency mode, the number is four; for the high-frequency model, it is two. In both modes, the vortex formation is phase-locked to the cylinder motion. For a specified mode with a fixed vortex number per cycle, the way the vortices evolve in the wake can be somewhat different by changing the vibration frequency and pitch ratio. These affecting factors have been examined in this work, and the associated vortex patterns have been characterized and compared.

10:43AM H12.00002 Characterization of vortex-induced vibration of a flexible cylinder , JESSICA SHANG, HOWARD STONE, ALEXANDER SMITS, Princeton University — In this study, the phenomena of 3D vortex-induced vibration (VIV) of a flexible cylinder (diameter D) is shown to be distinct from 2D VIV. We seek to identify correlations between wake regimes and vibration responses for a low mass-ratio (m^* = 1.2), flexible (E = 1.2 MPa, natural frequency in water f_N = 0.37 Hz) cantilevered cylinder undergoing cross-flow for reduced velocity U^* = 20-120 ($U^* = U/f_N D$). A P+S wake mode appears for a range of U^*; the onset of this range may be correlated with a hysteretic jump to an upper branch in the transverse amplitude response ($A_Y^* = A_Y/D$) at several locations along the midspan. This asymmetric wake mode does not present a unique transverse frequency response ($f_Y^* = f_Y/f_N$) in the cylinder. The upper branch in the amplitude response gives way to an abrupt decrease in A_Y^* to a lower branch, accompanied by a bifurcation in f_Y^*. The bifurcation takes place over a narrow range of U^* where the lower f_Y^* gradually transfers power to a higher f_Y^*, and may demarcate a wake transition regime between laminar and turbulence states.

10:56AM H12.00003 In-line and cross-flow multi-frequency vortex-induced vibrations of a long flexible cylinder are phase-locked under wake-body synchronization , REMI BOURGUET, IMFT - CNRS, GEORGE KARNIADAKIS, Brown University, MICHAEL TRIANTAFYLLOU, MIT — A slender flexible body with bluff cross-section immersed in cross-flow exhibits vortex-induced vibrations. The vibrations are excited by the flow under a condition of lock-in defined as the synchronization between vortex formation and body displacement. Within a sheared current, the possible occurrence of the lock-in condition at a number of different locations can lead to broadband vibrations involving a wide range of excited frequencies and structural wavenumbers. In a previous study focusing on the vortex-induced vibrations of a flexible cylinder at a single frequency in each direction, we have found that the lock-in condition is established through counter-clockwise figure-eight trajectories where the body moves upstream at the extremes of the cross-flow oscillation. In the present work, on the basis of direct numerical simulation results, we show that this mechanism can be generalized to multi-frequency responses: even if the trajectory shape substantially departs from a figure eight, the phase difference between the components of the in-line and cross-flow vibrations locally involved in the lock-in phenomenon remains within a particular range, associated with counter-clockwise figure-eight orbits in the mono-frequency case.

11:09AM H12.00004 Enhancing Vortex Induced Vibration of a Circular Cylinder by Using Roughness Strips[1] , ASHWIN VINOD, Missouri University of Science & Technology, Rolla, MO, ARINDAM BANERJEE, Lehigh University, Bethlehem, PA — The current experimental work focuses on studying the effects of surface roughness on vortex induced vibration (VIV) of an elastically mounted circular cylinder which is free to vibrate in a direction transverse to the flow. Our objective is to identify configurations which lead to high amplitudes of vibrations and a greater range of synchronization that can be successfully used for energy harvesting. Different configurations such as smooth cylinders, cylinder with zero roughness strips, and prescribed roughness (using sand paper) were used. Experiments were also conducted with the zero roughness strips at different angles around the cylinder to verify the effect of the position of the strip. All results were also found to be dependent on the spring stiffness. Variations were observed in the amplitude and frequency response profiles for the different cases investigated.

[1]The authors acknowledge support of the Office of Naval Research (Grant # ONR-N000141210495, Program Manager: Ronald Joslin)

11:22AM H12.00005 Use of targeted energy transfer to delay Kármán vortex shedding and suppress vortex-induced vibration in flow past a cylinder , RAVI K.R. TUMKUR, RAMON E. CALDERER, LAWRENCE A. BERGMAN, ALEXANDER F. VAKAKIS, ARIF MASUD, ARNE J. PEARLSTEIN, University of Illinois at Urbana-Champaign — For two-dimensional flow past a circular cylinder whose motion is constrained by a linear spring to be perpendicular to the mean flow, we report computations showing that "targeted energy transfer" using a nonlinear energy sink (NES; consisting of a mass, a linear damper, and an essentially nonlinear spring) not only can reduce the amplitude of the cylinder motion, but can also increase the critical Reynolds number (*Re*) at which the Kármán vortex street first appears. Absent the NES, the critical *Re* at which vortex shedding (and hence cylinder motion) sets in is determined as a function of the stiffness of the linear spring. Over a wide range of stiffness, the NES is shown to delay the onset to higher *Re*.

11:35AM H12.00006 Using LCS to identify vortex shedding on a cylinder in cross-flow , MELISSA GREEN, Syracuse University — The transition from steady separation to unsteady vortex shedding downstream of a circular cylinder in cross-flow is examined using a Lagrangian coherent structure analysis. Velocity data is gathered from both 2D and 3D simulations at a Reynolds number shortly after transition ($Re = 100$). At transition, when flow begins to entrain into and detrain from the cylinder wake, the wake as described using LCS undergoes a distinct qualitative change. This event in the evolution of the LCS will offer new information about possible timing and location at which to implement effective flow control to mitigate the shedding and unsteady forces on the cylinder body.

11:48AM H12.00007 Experiments on the flow around yawed and fixed cylinder: Forces and Flow measurements[1] , GUILHERME R. FRANZINI, RAFAEL S. GIORIA, IVAN KORKISCHKO, JULIO R. MENEGHINI, ANDRE L.C. FUJARRA, University of São Paulo — Flow around yawed and fixed circular cylinders were experimentally investigated at a Recirculating Water Channel facility. The total hydrodynamic loads were measured by using a 6DOF load cell and flow measurements were carried out using the 2D PIV technique. The cylinders were yawed in angles up to 45 degrees for both upstream and downstream orientations. For all the experiments, the aspect ratio is close to 13 and the lower end of the model is kept close to the channel floor. For the force measurements, the Reynolds number considering only the component normal to the cylinder axis lies in the interval $4000 < \mathrm{Re}_n < 14000$. The PIV measurements were carried out at Re_n to 9000. Differences were observed in the force coefficients plots depending on the orientation. Low frequencies component $\mathrm{fD/U}_n < 0.08$ were observed in lift force spectra in the case of upstream orientation. The Koopman decomposition applied to the PIV snapshots helped to a better understanding of the results.

[1]The authors acknowledge financial support from FAPESP, FINEP and CNPq.

12:01PM H12.00008 Flow around an inclined cylinder with different end plates boundary conditions[1] , RAFAEL GIORIA, NDF, Dept. Mech. Eng., Escola Politecnica of University of Sao Paulo, Brazil, GUILHERME FRANZINI, Dept. Naval Eng., Escola Politecnica of University of Sao Paulo, Brazil, JULIO MENEGHINI, NDF, Dept. Mech. Eng., Escola Politecnica of University of Sao Paulo, Brazil, ANDRE FUJARRA, Dept. Naval Eng., Escola Politecnica of University of Sao Paulo, Brazil — Numerical simulations of the flow around an inclined circular cylinder are carried out using the Lattice Boltzmann Method with Reynolds number Re=9,000 based on the normal component of the free-stream. The wall at the tips of the cylinder are modeled by two different boundary conditions: no-slip condition on the bottom wall and slip boundary conditions on the top wall. The resulting flows are different for an inclined cylinder with bottom tip upstream and another one with bottom tip downstream. This is related to the different boundary conditions on top and bottom. These differences are assessed via force coefficients comparison, near wake probes analysis and near wake visualization of the flow simulations. The main different feature observed is a streamwise vortical structure near the lower tip of the cylinder inclined upstream. This flow structure is not observed at the downstream orientation. The dominance of the upstream end of the cylinder in the wake and the asymmetry of the boundary conditions chosen for the walls seem to be responsible for the differences between upstream and downstream results.

[1]Authors ackowledge the support from Fapesp, Finep and CNPq.

12:14PM H12.00009 Reorientation of Vorticity on a Rapidly Accelerating Finite Aspect Ratio Plate , JOCHEN KRIEGSEIS, University of Calgary, MATTHIAS KINZEL, California Institute of Technology, DAVID RIVAL, University of Calgary — In recent studies the competition between the developing leading-edge vortex (LEV) and tip vortex (TV) has been considered from an Eulerian perspective. Such analyses are limited in that little is understood regarding the reorientation of vorticity layers from the attached boundary layers (BL). The vortex formation is fed by both the shear layers as well as the original BL vorticity. The purpose of the present work, therefore, is to uncover the influence of the BL on the formation of these vortices. 3D Particle Tracking Velocimetry (PTV) experiments have been performed so as to measure the flow around a low aspect ratio plunging flat plate. From the PTV results, Lagrangian structures have been identified that originate from the plate surface. The mass contribution of the boundary layer to the formation of the LEV and TV is discussed. Moreover, the reorientation of the BL vorticity during the process is tracked. By studying the reorientation of mass-containing vorticity, the close connection between the BL at the first instant of motion, and the salient vortices at later stages of the formation process is illustrated. Finally, the Lagrangian structures are compared with direct-force measurements to elucidate the influence of the BL vorticity distribution on the unsteady loadings.

Monday, November 19, 2012 10:30AM - 12:27PM –

Session H13 Rayleigh-Taylor Instability I 27A - Chair: Andrew Lawrie, University of Bristol

10:30AM H13.00001 Hydrodynamic instabilities and Boundary Value Problem , SNEZHANA I. ABARZHI, University of Chicago, Chicago, IL, USA — For the first time, on the basis of conservation principles and thermodynamics laws, we derive the generalized Rankine-Hugoniot conditions that can be applied for unsteady and curved fronts. The conditions describe the dynamics of the interface (front) in an explicit and covariant form and can be employed in convergent or Cartesian system of coordinates for three-dimensional systems. The theoretical framework is applied to the instabilities of Landau-Darrieus (LD), classical Rayleigh-Taylor (RT) and ablative Rayleigh-Taylor (ART). It is shown that in the case when if there is mass flux across the interface and no acceleration (LD), the front can be unstable only if the energy flux across the front is imbalanced. When acceleration is present (RT and ART), the dependence is obtained of the instability growth-rate on the mass flow and energy imbalance across the front. Connection between the ablative RTI and classical RTI is made. The stabilization mechanisms are discussed. The obtained results provide a theoretical framework for design of experiments under conditions relevant to inertial confinement fusion.

10:43AM H13.00002 Direct Numerical Simulations of Rayleigh-Taylor instability with gravity reversal , DANIEL LIVESCU, Los Alamos National Laboratory, TIE WEI, New Mexico Tech — In order to study the variable acceleration effects on the development of Rayleigh-Taylor instability (RTI), two unit problems are proposed: reversing the gravity and setting the gravity to zero in the turbulent stage of classical RTI. Data from high resolution Direct Numerical Simulations, covering the range of Atwood numbers from 0.04 to 0.9, are used to examine the modifications in the layer structure and turbulence properties following the change in gravity. After gravity reversal, the density inversion regions lead to new local RTI development, which efficiently mixes the large scales of the flow. This also introduces a strong directionality in the alignment of vorticity and strain rate eigenvectors. In addition, the turbulent transport reacts much faster to the change in gravity compared to the mean density. This renders the popular gradient diffusion hypothesis inappropriate for such flows, which pose significant challenges for engineering turbulence models.

10:56AM H13.00003 Numerical investigations of Rayleigh-Taylor instability development from an initially isotropic turbulent velocity field , POOYA MOVAHED, BRUCE FRYXELL, ERIC JOHNSEN, University of Michigan, Ann Arbor — The Rayleigh-Taylor instability (RTI) is a process by which the misalignment of the pressure and the density gradient at unstably stratified interfaces generates baroclinic vorticity. This process can transition from laminar flow to a fully turbulent mixing region. Numerical simulations of RTI are traditionally initialized by either perturbing the density field at the interface or by transforming the density perturbations to velocity perturbations using linear theory. In this study, the initial interface separates the light and heavy fluids in an existing isotropic turbulent velocity field extending in the whole domain in each fluid. These initial conditions enable us to reach high Reynolds numbers rapidly during the simulation. First, we neglect gravity and quantify isotropy and intermittency of the decaying turbulent field in the mixing region. Second, the problem is revisited in the presence of a gravitational field. The initial fluctuating velocity field perturbs the interface and the baroclinic vorticity generated in the mixing region due to the instability provides energy for the initial decaying turbulent field. A comparison of relevant physical quantities regarding isotropy and mixing is made to the first case. The simulations are performed using a high-order accurate minimally dissipative kinetic-energy preserving and interface capturing scheme. This research was supported in part by the DOE NNSA under the Predictive Science Academic Alliance Program by grant DEFC52-08NA28616.

11:09AM H13.00004 Efficient mixing in stratified flows: Rayleigh-Taylor instability within a stable stratification , MEGAN DAVIES WYKES, STUART DALZIEL, University of Cambridge — Turbulent mixing is generated at the Rayleigh-Taylor unstable interface between two stably stratified layers. Measurements of the density profile in a salt stratified laboratory experiment, before and after this process, show very high mixing efficiencies of 0.6 to 0.8. This is significantly higher than that seen in either shear flows or classical two-layer Rayleigh-Taylor instability. We discuss these startling results and present a simple turbulent diffusion model that captures the essential dynamics of the flow.

11:22AM H13.00005 Estimates of molecular mixing in confined Rayleigh-Taylor instability , ANDREW LAWRIE, University of Bristol, STUART DALZIEL, University of Cambridge — We examine the behaviour of a system in which a RT unstable interface is confined between stable continuous stratifications. Recent experiments with linear stratifications (Lawrie & Dalziel 2011, JFM) indicate an intrinsic limit to a fluid's ability to mix, which here can be measured robustly between quiescent initial and final states. Standard incompressible ILES does not match well because it cannot respect the balance of energy conversions observed in experiment. ILES operates with Sc=O(1), whereas Sc=700 in our experiments. Lawrie & Dalziel detailed the relation between the p.d.f. of the density field and the availability of energy in the system. Here we extract the evolution of the p.d.f. over the life-cycle of the instability, and thus quantify the ILES mixing estimates in both 2D and 3D RT cases. In 3D, energy cascades to small scales, so the stretching of material surfaces that it induces tends to occur at comparable scales and this is the optimal condition for doing mixing. In 2D, however, energy accumulates at large scales and thus material surfaces do not become so rapidly stretched. We view the 2D case as an analogue for high Schmidt number behaviour, and this helps us understand the modelling approximations in 3D cases.

11:35AM H13.00006 Evaluation of the Predictions of a Four-Equation Reynolds-Averaged Navier-Stokes Model Applied to Rayleigh-Taylor Instability-Induced Mixing[1] , KYLE K. MACKAY, Utah State University, OLEG SCHILLING, Lawrence Livermore National Laboratory — An implicit-in-time multicomponent, weighted essentially nonoscillatory implementation of a four-equation K-ϵ based Reynolds-averaged Navier–Stokes model is used to simulate Rayleigh–Taylor turbulent mixing at Atwood numbers ranging from 0.05–0.9. The mechanical turbulence equations are coupled to modeled transport equations for the scalar variance and its dissipation rate. The predicted evolution of the mixing layer, molecular mixing and other quantities are compared to available experimental data, as well as to analytical self-similar solutions. The predictive capability of the model is evaluated, and several parametric studies are also presented.

[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.

11:48AM H13.00007 Large Eddy Simulations of the Tilted Rig Experiment: A Two-dimensional Rayleigh-Taylor Instability Case[1] , BERTRAND ROLLIN, NICHOLAS A. DENISSEN, JON M. REISNER, MALCOLM J. ANDREWS, Los Alamos National Laboratory — The tilted rig experiment is a derivative of the rocket rig experiment designed to investigate turbulent mixing induced by the Rayleigh-Taylor (RT) instability. A tank containing two fluids of different densities is accelerated downwards between two parallel guiding rods by rocket motors. The acceleration is such that the pressure and density gradients face opposite directions at the fluids interface, creating a Rayleigh-Taylor unstable configuration. The rig is tilted such that the tank is initially at an angle and the acceleration is not perpendicular to the fluids interface when the rockets fire. This results in a two dimensional Rayleigh-Taylor instability case where the fluids experience RT mixing and a bulk overturning motion. The tilted rig is therefore a valuable experiment to help calibrating two-dimensional mixing models. Large Eddy Simulations of the tilted rig experiments will be compared to available experimental results. A study of the behavior of turbulence variables relevant to turbulence modeling will be presented. LA-UR 12-23829.

[1]This work was performed for the U.S. Department of Energy by Los Alamos National Laboratory under Contract No.DEAC52- 06NA2-5396.

12:01PM H13.00008 RANS Simulations of the Tilted Rig Experiment:A Two-dimensional Rayleigh-Taylor Instability Case , NICHOLAS DENISSEN, BERTRAN ROLLIN, JON REISNER, MALCOLM ANDREWS, Los Alamos National Laboratory — Modeling turbulent mixing due to unstable density stratification is of fundamental interest in many multiphysics applications. RANS models remain the tool of choice for efficient estimates of the effects of turbulence on complex problems. While many RANS models have been validated for canonical Rayleigh–Taylor turbulence, applications of interest often have non-planar/dynamic interfaces. This presentation will address the ability of a multispecies, compressible, turbulence model to compute RT mixing on a moving interface. The simulations are based on the tilted rocket-rig experiments designed to study mixing of fluids by the Rayleigh-Taylor instability. In this experiment, a tank containing two fluids of different densities is accelerated downward with the rig inclined by a few degrees off vertical. The RANS simulations are be compared to experiments, direct numerical simulations and large eddy simulations to analyze the model's ability to capture 2D flow features.

12:14PM H13.00009 The Dynamics of Rayleigh-Taylor Stable and Unstable Contact Discontinuities with Anisotropic Thermal Conduction[1] , DANIEL LECOANET, IAN PARRISH, ELIOT QUATAERT, University of California - Berkeley — We study the effects of anisotropic thermal conduction along magnetic field lines on an accelerated contact discontinuity in a weakly collisional plasma. Anisotropic conduction can result in doubly-diffusive instabilities, including the magnetothermal instability (MTI) and the heat flux driven buoyancy instability (HBI). We run fully non-linear numerical simulations of a contact discontinuity with anisotropic conduction. The non-linear evolution can be described as a superposition of three physical effects: temperature diffusion due to vertical conduction, the Rayleigh-Taylor instability (RTI) and the HBI. In simulations with RTI-stable contact discontinuities, the temperature discontinuity spreads due to vertical heat conduction. The HBI slows this temperature diffusion by reorienting initially vertical magnetic field lines to a more horizontal geometry, eventually stopping vertical temperature diffusion. In simulations with RTI-unstable contact discontinuities, the dynamics are initially governed by temperature diffusion, but the RTI becomes increasingly important at late times. These results could be important in various astrophysical contexts including supernova remnants, solar prominences and cold fronts in galaxy clusters.

[1]DL is supported by the Hertz Foundation and NSF Grant DGE 1106400; IP & EQ are supported in part by NASA Grant ATP09-0125, NSF-DOE Grant PHY-0812811, and by the David and Lucille Packard Foundation.

Monday, November 19, 2012 10:30AM - 12:27PM –

Session H14 General Experiments III 27B - Chair: Jun Chen, Purdue University

10:30AM H14.00001 Validity of Molecular Tagging Velocimetry in a Cavitating Flow for Turbopump Analysis , KAYLA KUZMICH, DOUG BOHL, Clarkson University — This research establishes multi-phase molecular tagging velocimetry (MTV) use and explores its limitations. The flow conditions and geometry in the inducer of an upper stage liquid Oxygen (LOX)/LH2 engine frequently cause cavitation which decreases turbopump performance. Complications arise in performing experiments in liquid hydrogen and oxygen due to high costs, high pressures, extremely low fluid temperatures, the presence of cavitation, and associated safety risks. Due to the complex geometry and hazardous nature of the fluids, a simplified throat geometry with water as a simulant fluid is used. Flow characteristics are measured using MTV, a noninvasive flow diagnostic technique. MTV is found to be an applicable tool in cases of low cavitation. Highly cavitating flows reflect and scatter most of the laser beam disallowing penetration into the cavitation cloud. However, data can be obtained in high cavitation cases near the cloud boundary layer. Distribution A: Public Release, Public Affairs Clearance Number: 12654

10:43AM H14.00002 An Approach for Correcting the Velocity Bias Error in One-Component Molecular Tagging Velocimetry: Theoretical Analysis[1] , AHMED NAGUIB, PATRICK HAMMER, SHAHRAM POUYA, MANOOCHEHR KOOCHESFAHANI, Michigan State University — One-component Molecular Tagging Velocimetry (1c-MTV) using line tagging has the advantage of providing velocity information with very high spatial resolution: at every pixel along the imaged line of tagged molecules. This renders the technique particularly suitable for near-wall, boundary-layer-resolved measurements. However, a non-negligible bias error could affect the measurements if the velocity component along the "tag line" is substantial relative to the measured component and/or when the time delay between image pairs is large. In this presentation, which is the first of a sequence of two talks, a theoretical analysis is detailed for expressing this bias error in a Taylor-series framework. The error estimate based on this analysis is validated using simple flows with known analytical solutions. Moreover, by truncating the Taylor-series expansion at a low order, it is possible to use the resulting form as basis for a practical method for linear, or higher-order, correction of the bias error. Experimental demonstration of the viability of the method will be presented in a follow-up talk.

[1]Supported by AFOSR grant FA9550-10-1-0342

10:56AM H14.00003 An Approach for Correcting the Velocity Bias Error in One-Component Molecular Tagging Velocimetry: Experimental Demonstration[1] , PATRICK HAMMER, SHAHRAM POUYA, AHMED NAGUIB, MANOOCHEHR KOOCHESFAHANI, Michigan State University — Measurements and simulated experiments have been performed to demonstrate the validity of a method to correct for the velocity bias error associated with single-component Molecular Tagging Velocimetry due to the presence of a velocity component along the line of tagged molecules. The method, the theoretical basis for which is presented in a preceding talk, utilizes two MTV images obtained using two different time delays relative to the un-delayed image to correct for the bias error. An experiment based on fully developed flow in a rectangular channel is used to validate the correction approach. In the experiment, a velocity component along the "tag line" is deliberately imposed by tagging molecules along a line that is inclined to the flow direction in the channel. The error produced by interrogation normal to the line is corrected using the new approach, and the result is compared against bias-error-free measurements based on interrogation of the line in the flow direction. The results show that the new approach successfully corrects for the bias error. The sensitivity of the correction to parameters such as measurement noise, time-delay pairs, and flow angle, is assessed using simulated experiments.

[1]Supported by AFOSR grant FA9550-10-1-0342

11:09AM H14.00004 Error Reduction in Molecular Tagging Velocimetry (MTV) Processing Using Image Filtering , DOUG BOHL, MIKE CASO, Clarkson University — Prior work has shown that the error level in MTV measurements is closely tied to the image Signal to Noise (SN) level. In practice the SN ratio will depend on experimental conditions such as attenuation, Field of View, laser power, camera, etc.; however, there is a minimum SN level that can be achieved for any given experiment. Experience has shown that MTV images typically have a SN=2-8. It is therefore desirable to be able to lower image noise after the images are acquired to reduce measurement error. In this work post processing MTV images using standard image filtering schemes such as Gaussian Blur, FFT (band pass), median filtering etc. were investigated. Past results on synthetic images showed that for very noisy images (i.e. SN<4) all filtering techniques improved the displacement error by 10-40%. As the SN increased filtering because less effective in decreasing error and in some cases increased the measurement error. The current work investigates the effect of image filtering on experimental images and compares those results to the synthetic image study.

11:22AM H14.00005 Voltage-Current Characteristics of Plasma Pressure Sensor[1] , ERIC MATLIS, THOMAS CORKE, CURTIS MARSHALL, University of Notre Dame, SIVARAM GOGINENI, Spectral Energies, LLC, UNIVERSITY OF NOTRE DAME TEAM, SPECTRAL ENERGIES TEAM — A pressure sensor based on the use of plasma as the sensing element is being developed. This is an AC-driven, continuous-wave plasma which is encapsulated between two metallic bare electrodes with a small air gap on the order of 0.03 mm. The sensor uses a non-equilibrium discharge at less than 20 Watts of power. This devices features an amplitude modulated carrier to measure both mean and dynamic pressure. The frequency response is limited only by the carrier frequency which can be as high as 1 MHz. Glow-to-Arc transition is controlled with the use of a capacitive and resistive circuit in series with the discharge. A pressure chamber is used to document the plasma power characteristics as the ambient pressure is controlled from atmospheric to 100 psi. Plasma power is controlled so as to maintain the plasma in the normal and abnormal glow regimes. The phase angle between voltage and current is recorded as a function of pressure. This analysis will aid in the development of a feedback control and calibration of the pressure sensor.

[1]NavAir SBIR

11:35AM H14.00006 Global Pressure Measurement of Unsteady-State Flow and Motion on Fluttering Airfoil , TAIKA OKABE, TAKESHI MIYAZAKI, The University of Electro-communications, KENICHI SAITOH, HIROTAKA SAKAUE, Japan Aerospace Exploration Agency — Pressure-sensitive paint (PSP) measurement has been applied to a fluid dynamic measurement. It can be applied to a steady-state flow in the transonic and supersonic wind tunnels. To extend the PSP measurement, an unsteady-state measurement is paid attention. It can be categorized by the unsteady-state flow and unsteady motion of the PSP-coated model. The former can be captured by using a porous PSP. The fastest PSP gives the response time on the order of ten microseconds. For the latter, the motion-capturing PSP method is studied to capture the unsteady motion. It consists of a two-color PSP and color camera. One color corresponds to the pressure-independent luminescence, and the other to the pressure-dependent luminescence. The former is used to cancel the pressure-independent distribution of a pressure-dependent image. The two-color images are simultaneously captured by the color camera. A combination of the unsteady-state flow and the motion can be occurred for an unsteady-state measurement. We present one of the cases, which is a flutter on an airfoil. By combining a porous PSP and the motion-capturing method, a pressure distribution on a fluttering airfoil (2-D YXX 30% span) is captured.

11:48AM H14.00007 Performance of an untethered micro-optical pressure sensor[1] , TINDARO IOPPOLO, MAURIZIO MANZO, PAUL KRUEGER, Southern Methodist University — We present analytical and computational studies of the performance of a novel untethered micro-optical pressure sensor for fluid dynamics measurements. In particular, resolution and dynamic range will be presented. The sensor concept is based on the whispering galley mode (WGM) shifts that are observed in micro-scale dielectric optical cavities. A micro-spherical optical cavity (liquid or solid) is embedded in a thin polymeric sheet. The applied external pressure perturbs the morphology of the optical cavity leading to a shift in its optical resonances. The optical sensors are interrogated remotely, by embedding quantum dots or fluorescent dye in the micro-optical cavity. This allows a free space coupling of excitation and monitoring of the optical modes without the need of optical fibers or other cabling. With appropriate excitation and monitoring equipment, the micro-scale sensors can be distributed over a surface (e.g., including flexible biological surfaces) to monitor the local pressure field.

[1]We acknowledge the financial support from the National Science Foundation through grant CBET-1133876 with Dr. Horst Henning Winter as the program director.

12:01PM H14.00008 Simultaneous measurement of morphological shape and 3D motion of objects using digital holographic microscopy , KYUNG WON SEO, SANG JOON LEE, Pohang University of Science and Technology — The phase-imaging technique using off-axis holography and the volumetric measurement of particle fields using in-line holography have been investigated separately. In the present study, we combine the advantages of these two techniques. A high-speed off-axis holographic system is established to simultaneously measure the morphology and the 3D motion of particles in a microtube flow with high spatial and temporal resolutions. The off-axis holography setup with transmission-type configuration is based on the principle of Mach–Zehnder interferometry. Off-axis holograms were numerically reconstructed in amplitude and phase, after frequency-domain filtering by adopting the angular spectrum method. The amplitude information of the reconstructed image indicates the 3D positions of the suspended particles, and phase information provides the object morphology or physical thickness. As a result, the 3D trajectories, instantaneous velocity, and the 3D shape of particles are extracted through a position detection and particle tracking velocimetry (PTV) algorithm and quantitative phase map. This holographic microscopy technique demonstrates the feasibility of the simultaneous measurement of the 3D dynamic behavior and temporal shape of objects.

12:14PM H14.00009 Automatic characterization of particle fields using digital holography , JIAN GAO, JUN CHEN, Purdue University, DANIEL GUILDENBECHER, PHILLIP REU, Sandia National Laboratories — An automatic algorithm is developed to characterize sizes and 3D positions of particle fields using digital holography. The reconstructed intensity image is used in conjunction with the edge sharpness of the particle image to automatically determine the level for thresholding of the intensity image. The morphology and transverse position of an individual particle are extracted from the binary image while its axial position is decided by maximizing the edge sharpness of the particle along the axial direction. A comparison using synthetic holograms with published particle detection approaches demonstrates the superiority of the proposed method. We further apply the method to segment a reconstructed image of a particle field. In particular, measurement errors introduced by transversely overlapped particles in dense particle fields are eliminated using a particle refinement algorithm based on 3D segmentation. The results demonstrate significant improvements in both the size and position measurements. The new method is applied to processing of experimental holograms of a particle field to further demonstrate its effectiveness.

Monday, November 19, 2012 10:30AM - 12:27PM –

Session H15 Biofluids: Animals 28A - Chair: John Bush, Massachusetts Institute of Technology

10:30AM H15.00001 Drinking strategies in nature , WONJUNG KIM, JOHN BUSH, MIT — We examine the fluid mechanics of drinking in nature. We classify the drinking strategies of a broad range of creatures according to the principal forces involved, and present physical pictures for each style. Simple scaling arguments are developed and tested against existing data. While suction is the most common drinking strategy, various alternative styles have evolved among creatures whose morphological, physiological, and environmental constraints preclude it. Particular attention is given to creatures small relative to the capillary length, whose drinking styles rely on relatively subtle interfacial effects.

10:43AM H15.00002 Numerical and experimental hydrodynamic analysis of suction cup bio-logging tag designs for marine mammals , MARK MURRAY, United States Naval Academy, ALEX SHORTER, Woods Hole Oceanographic Institution, LAURENS HOWLE, Duke University, MARK JOHNSON, University of St Andrews, MICHAEL MOORE, Woods Hole Oceanographic Institution — The improvement and miniaturization of sensing technologies has made bio-logging tags, utilized for the study of marine mammal behavior, more practical. These sophisticated sensing packages require a housing which protects the electronics from the environment and provides a means of attachment to the animal. The hydrodynamic forces on these housings can inadvertently remove the tag or adversely affect the behavior or energetics of the animal. A modification to the original design of a suction cup bio-logging tag housing was desired to minimize the adverse forces. In this work, hydrodynamic loading of two suction cup tag designs, original and modified designs, were analyzed using computational fluid dynamics (CFD) models and validated experimentally. Overall, the simulation and experimental results demonstrated that a tag housing that minimized geometric disruptions to the flow reduced drag forces, and that a tag housing with a small frontal cross-sectional area close to the attachment surface reduced lift forces. Preliminary results from experimental work with a common dolphin cadaver indicates that the suction cups used to attach the tags to the animal provide sufficient attachment force to resist failure at predicted drag and lift forces in 10 m/s flow.

10:56AM H15.00003 Modeling huddling penguins , FRANCOIS BLANCHETTE, AARON WATERS, ARNOLD KIM, University of California Merced — We present a model of the behavior of huddling penguins. We focus on the densest huddles, formed during storms, where penguins may be considered to leave no openings in the interior of the huddle. We compute a temperature distribution around the huddle, accounting for the effects of the wind. The dynamics of the huddle are based on an iterative process where the most exposed penguin relocates to the most sheltered location available. We study the effects of wind strength, number of penguins, and random perturbations. We find that our model produces huddles that agree qualitatively with actual huddles in terms of shape and downwind displacement. Moreover, the exposure to the wind appears to be shared nearly equally among penguins, despite the fact that our model assumes only that each penguin aims to minimize its own heat loss.

11:09AM H15.00004 Robot locomotion on weak ground , FEIFEI QIAN, CHEN LI, Georgia Institute of Technology, PAUL UMBANHOWAR, Northwestern University, DANIEL GOLDMAN, Georgia Institute of Technology — Natural substrates like sand, soil, and leaf litter vary widely in penetration resistance. Little is known about how animals (and increasingly robots) respond to this variation. To address this deficit, we built an air fluidized bed trackway, in which we control penetration resistance of 1 mm granular substrates down to zero by increasing the upward flow rate, Q, to the fluidization transition. Using a 2.5 kg bio-inspired hexapedal robot as our model locomotor, we systematically study how locomotion performance (average forward speed, v) varies with penetration resistance, limb kinematics, and foot morphology. Average robot speed decreases with increasing Q, and decreases faster for robots with higher leg frequency or narrower leg width. A previously developed model, which captured the robot's performance on granular media with $Q = 0$, also captures the observed performance for weakened states with $Q > 0$. A single dimensionless control parameter from the model, which combines gait and ground parameters, determines v for all penetration resistances. Our ground control technique and modeling approach provide a way to probe and understand the limits of locomotor performance on yielding substrates.

11:22AM H15.00005 How does a Tiger beetle catches its prey? , Z. JANE WANG, Physics and MAE, Cornell University, ANDREAS HASELSTEINER, University of Bremen, COLE GILBERT, Entomology, Cornell University — When a beetle chases its prey, what laws does it follow, if any? Theoretically, there are multiples strategies that a beetle could use to intercept its prey. By analyzing the pursuit dynamics of Tiger beetles, we found that the beetle adjusts its orientation to minimize the angle between its heading and the prey. The adjustment is linearly proportional the error angle with a time delay. We offer a mechanical explanation of this correlation between the angle and the angular rotation of the beetle. We further suggest an physical interpretation for the time delay constant.

11:35AM H15.00006 Impact and intrusion of the foot of a lizard running rapidly on sand , CHEN LI, University of California, Berkeley, TONIA HSIEH, Temple University, PAUL UMBANHOWAR, Northwestern University, DANIEL GOLDMAN, Georgia Institute of Technology — The desert-dwelling zebra-tailed lizard (*Callisaurus draconoides*, 10 cm, 10 g) runs rapidly ($\sim$10 BL/s) on granular media (GM) like sand and gravel. On loosely packed GM, its large hind feet penetrate into the substrate during each step. Based on above-ground observation, a previous study (Li et al., JEB 2012) hypothesized that the hind foot rotated in the vertical plane subsurface to generate lift. To explain the observed center-of-mass dynamics, the model assumed that ground reaction force was dominated by speed-independent frictional drag. Here we use x-ray high speed video to obtain subsurface foot kinematics of the lizard running on GM, which confirms the hypothesized subsurface foot rotation following rapid foot impact at touchdown. However, using impact force measurements, a resistive force model, and the observed foot kinematics, we find that impact force during initial foot touchdown and speed-independent frictional drag during rotation only account for part of the required lift to support locomotion. This suggests that the rapid foot rotation further allows the lizard to utilize inertial forces from the local acceleration of the substrate (particles), similar to small robots running on GM (Qian et al., RSS 2012) and the basilisk (Jesus) lizard running on water.

11:48AM H15.00007 Ground resistance influences lizard burial in dry and wet sand , SARAH SHARPE, ROBYN KUCKUK, DANIEL GOLDMAN, Georgia Institute of Technology — Many terrestrial animals move within soil in which water content can vary, and little is known about how water content affects locomotor performance. To investigate the effect of water content on burial, we created controlled dry and wet substrates. We used 0.3 mm glass particles and varied water content W, the mass of water to mass of dry loosely packed sand. Drag force on a submerged 1.6 cm diameter rod increased by a factor of 4 as W increased from 0 to 0.03, after which force increases were small. Drag force in wet media periodically fluctuated with time and corresponded with surface fracturing. We characterized how W affected burial performance and strategy of a generalist burrower, the ocellated skink lizard (*Chalcides ocellatus*). High speed x-ray imaging was used to measure head, body and limb kinematics in substrates with W= 0 and W= 0.03. In both states during burial the body was maintained in a curved posture and the animal moved using a start-stop motion. During movement, the head oscillated and the forelimb on the convex side of the body was used to push the animal forward. Both speed and angular excursion of the head oscillation decreased in the W= 0.03 state. The differences in locomotion were attributed to the changing resistance force within the media.

12:01PM H15.00008 ABSTRACT WITHDRAWN –

12:14PM H15.00009 Jumping of water striders on water , EUNJIN YANG, JAEHAK SON, PIOTR JABLONSKI, HO-YOUNG KIM, Seoul National University — Small insects such as water striders, springtails, fishing spiders freely move on water by adopting various modes of locomotion, such as rowing, galloping, jumping and meniscus-climbing. As the physics of jumping have not yet been fully understood among those ways of semi-aquatic propulsion, here we present the results of a combined experimental and theoretical investigation of the dynamics of water striders leaping off water. We first image and analyze the trajectories of the legs and body of jumping water striders of three different species with a high-speed camera. We then theoretically compute the forces acting on the body by considering the capillary interaction between the flexible legs and deforming water meniscus. Our theory enables us to predict the maximum take-off speed for given leg lengths. The experimental measurements suggest that the water striders drive their legs near the optimal speed to gain the maximum take-off speed.

Monday, November 19, 2012 10:30AM - 12:27PM –

Session H16 Biofluids: Heart Ventricles and Assist Devices 28B - Chair: Rajat Mittal, Johns Hopkins University

10:30AM H16.00001 Simulating Pediatric Ventricular Assist Device Operation Using Fluid Structure Interaction , CHRIS LONG, YURI BAZILEVS, ALISON MARSDEN, University of California, San Diego — Ventricular Assist Devices (VADs) provide mechanical circulatory support to patients in heart failure. They are primarily used to extend life until cardiac transplant, but also show promise as a "bridge-to-recovery" device in pediatric patients. Commercially available pediatric pumps are pulsatile displacement pumps, with two distinct chambers for air and blood separated by a thin, flexible membrane. The air chamber pneumatically drives the membrane, which drives blood through the other chamber via displacement. The primary risk factor associated with these devices is stroke or embolism due to thrombogenesis in the blood chamber, occurring in as many as 40% of patients. Our goal is to perform simulations that accurately model the hemodynamics of the device, as well as the non-linear membrane buckling. We apply a finite-element based fluid solver, with an Arbitrary Lagrangian-Eulerian (ALE) framework to account for mesh motion. Isogeometric Analysis with a Kirchhoff-Love shell formulation is used on the membrane, and two distinct fluid subdomains are used for the air and blood chambers. The Fluid Structure Interaction (FSI) problem is solved simultaneously, using a Matrix Free method to model the interactions at the fluid-structure boundary. Methods and results are presented.

10:43AM H16.00002 Heart Rate and AV delay modify left ventricular filling vortex properties[1] , JUAN C. DEL ALAMO, MAE Dept UC San Diego, La Jolla, CA, YOLANDA BENITO, JAVIER BERMEJO, Cardiology Dept, Hospital Gregorio Maranon, Madrid, Spain, MARTA ALHAMA, Internal Medicine Dept, Scripps Green Hospital, La Jolla, CA, RAQUEL YOTTI, CANDELAS PEREZ DEL VILLAR, Cardiology Dept, Hospital Gregorio Maranon, Madrid, Spain, PABLO MARTINEZ-LEGAZPI, MAE Dept UC San Diego, La Jolla, CA, ANA GONZALEZ MANSILLA, FRANCISCO FERNANDEZ-AVILES, Cardiology Dept, Hospital Gregorio Maranon, Madrid, Spain — Intraventricular flow generates a vortex ring during rapid filling that optimizes filling, couples inflow kinetic energy to ejection, improves blood mixing and avoids stasis. LV vorticity has been related to chamber geometrical properties, but the effects of electrical events have never been characterized, partly due to the difficulty of performing MRI in patients with implanted devices. We have recently developed a new method that allows measuring vortex properties by processing conventional transthoracic color-Doppler sequences. Using this modality, 27 patients carrying an implantable cardiac resynchronization device were studied after AV optimization at 100 beats per minute. Our results reveal that, compared to optimal AV, the main vortex component remained closer to the base during 100BPM (difference = -20% of Lax length, $p< .05$) and closer to the apex when AV is minimized (diff= +11% of Lax, $p< .05$). Radius, circulation and energy of the vortices were larger when AV is maximized ($p< .05$). In conclusion, the duration of diastole, as modulated by heart rate and AV-delay, significantly modifies intraventricular vortex dynamics.

[1]Funded by NIH Grant R21HL108268

10:56AM H16.00003 On the clinical characterization of impulse and suction force contributions by the diastolic left ventricular vortex[1] , PABLO MARTINEZ-LEGAZPI, MAE Dept UC San Diego, La Jolla, CA, MARTA ALHAMA, Dept of Internal Medicine, Scripps Green Hospital, La Jolla CA, YOLANDA BENITO, JAVIER BERMEJO, RAQUEL YOTTI, ESTHER PEREZ-DAVID, ALICIA BARRIO, CANDELAS PEREZ-DEL-VILLAR, ANA GONZALEZ-MANSILLA, FRANCISCO FERNANDEZ-AVILES, Dept of Cardiology, Hospital Gregorio Maranon, Madrid, Spain, JUAN C. DEL ALAMO, MAE Dept UC San Diego, La Jolla, CA — One of the fluid-dynamic mechanisms that characterize the diastolic phase of the cardiac cycle is the formation of a left ventricular (LV) vortex ring that has been proposed to improve LV filling. However, direct clinical quantification of the contribution of this vortex to LV filling is elusive. In this clinical study, we considered 20 patients with dilated cardiomyopathy (DCM) and 40 healthy volunteers. We have developed and validated a method that derives two-dimensional maps of the LV flow from standard color-Doppler sequences. This study employs the new imaging modality in combination with a vortex identification method and a panel method in order to isolate and estimate the direct contribution of the LV vortex to fluid impulse and suction force during filling in the healthy and diseased populations.

[1]Funded by NIH Grant R21HL108268

11:09AM H16.00004 High-resolution numerical simulation of Left Ventricular Hemodynamics Guided by in-vivo Cardiac Magnetic Resonance Data[1] , TRUNG LE, FOTIS SOTIROPOULOS, University of Minnesota, LUCIA MIRABELLA, BRANDON CHAFFINS, ARVIND SANTHANAKRISHNAN, Georgia Institute of Technology, JOHN OSHINSKI, Georgia Institute of Technology & Emory, AJIT YOGANATHAN, Georgia Institute of Technology, UNIVERSITY OF MINNESOTA COLLABORATION, GEORGIA INSTITUTE OF TECHNOLOGY COLLABORATION — We study the fluid dynamics within a patient-specific left ventricle (LV) during diastole using both numerical simulations and in-vivo data. The kinematics of the LV is reconstructed from high-resolution Magnetic Resonance Imaging (MRI) data acquired on a healthy volunteer, using image segmentation and a surface registration process. The flow velocity is acquired using phase-contrast MRI at the mitral orifice and at an additional parallel plane inside the ventricle. Numerical simulations are carried out using the CURVIB method (Ge et al., JCP, 2007) with the MRI reconstructed LV wall motion imposed as boundary condition. The numerical simulations show the highly dynamic environment of the flow field. The mitral vortex ring is formed during early diastolic filling and breaks down into small scale structures. The simulated hemodynamics are compared with phase-contrast MRI measurements and previous simulations in which the LV wall motion was obtained from a lumped parameter model (Le and Sotiropoulos, Eur. J. Mechanics B - Fluids, 2012)

[1]We acknowlege NIH Grant RO1-HL-07262 and the Minnesota Supercomputing Institute support.

11:22AM H16.00005 Right Ventricular Hemodynamics in Patients with Pulmonary Hypertension , JAMES BROWNING, University of Colorado Boulder, BRETT FENSTER, National Jewish Health and University of Colorado Denver, JEAN HERTZBERG, University of Colorado Boulder, JOYCE SCHROEDER, National Jewish Health and University of Colorado Denver — Recent advances in cardiac magnetic resonance imaging (CMR) have allowed for characterization of blood flow in the right ventricle (RV), including calculation of vorticity and circulation, and qualitative visual assessment of coherent flow patterns. In this study, we investigate qualitative and quantitative differences in right ventricular hemodynamics between subjects with pulmonary hypertension (PH) and normal controls. Fifteen (15) PH subjects and 10 age-matched controls underwent same day 3D time resolved CMR and echocardiography. Echocardiography was used to determine right ventricular diastolic function as well as pulmonary artery systolic pressure (PASP). Velocity vectors, vorticity vectors, and streamlines in the RV were visualized in Paraview and total RV Early (E) and Atrial (A) wave diastolic vorticity was quantified. Visualizations of blood flow in the RV are presented for PH and normal subjects. The hypothesis that PH subjects exhibit different RV vorticity levels than normals during diastole is tested and the relationship between RV vorticity and PASP is explored. The mechanics of RV vortex formation are discussed within the context of pulmonary arterial pressure and right ventricular diastolic function coincident with PH.

11:35AM H16.00006 Computational Modeling of the Effects of Myocardial Infarction on Left Ventricular Hemodynamics[1] , VIJAY VEDULA, JUNG HEE SEO, RAJAT MITTAL, Johns Hopkins University, STEFANIA FORTINI, GIORGIO QUERZOLI, University of Roma La Sapienza — Most in-vivo and modeling studies on myocardial infarction and ischemia have been directed towards understanding the left ventricular wall mechanics including stress-strain behavior, end systolic pressure-volume correlations, ejection fraction and stroke work. Fewer studies have focused on the alterations in the intraventricular blood flow behavior due to local infarctions. Changes in the motion of the endocardium can cause local circulation and stagnation regions; these increase the blood cell residence time in the left ventricle and may eventually be implicated in thrombus formation. In the present study, we investigate the effects of myocardial infarction on the ventricular hemodynamics in simple models of the left ventricle using an immersed-boundary flow solver. Apart from the Eulerian flow features such as vorticity and velocity flow fields, pressure distribution, shear stress, viscous dissipation and pump work, we also examine the Lagrangian dynamics of the flow to gain insights into the effect of flow dynamics on thrombus formation. The study is preceded by a comprehensive validation study which is based on an in-vitro experimental model of the left ventricle and this study is also described.

[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.

11:48AM H16.00007 A Methodology for Quantifying Heart Function in the Embryonic Zebrafish[1] , BRENNAN JOHNSON, DEBORAH GARRITY, LAKSHMI DASI, Colorado State University — Several studies have linked epigenetic factors such as blood flow dynamics and cardiac function to proper heart development. To better understand this process, it is essential to develop robust quantitative methods to investigate the blood dynamics and wall kinematics in vivo. Here, we develop a methodology that can be used throughout the early stages of development which requires no specialized equipment other than a bright field microscope and high-speed camera. We use the embryonic zebrafish as our model due to its superb optical access and widespread acceptance as a powerful model for human heart development. Using these methods, we quantify blood flow rates, stroke volume, cardiac output, ejection fraction, and other important parameters related to heart function. We also investigate the pumping mechanics from heart tube to looped configuration. We show that although the mechanism changes fundamentally, it does so in a continuous fashion that can incorporate combined pumping mechanisms at intermediate stages. This work provides a basis for quantitatively comparing normal and abnormal heart development, and may help us gain a better understanding of congenital heart defects.

[1]Funded by NSF

12:01PM H16.00008 Human Aorta Is a Passive Pump , NIEMA PAHLEVAN, MORTEZA GHARIB, California Institute of Technology — Impedance pump is a simple valveless pumping mechanism that operates based on the principles of wave propagation and reflection. It has been shown in a zebrafish that a similar mechanism is responsible for the pumping action in the embryonic heart during early stages before valve formation. Recent studies suggest that the cardiovascular system is designed to take advantage of wave propagation and reflection phenomena in the arterial network. Our aim in this study was to examine if the human aorta is a passive pump working like an impedance pump. A hydraulic model with different compliant models of artificial aorta was used for series of in-vitro experiments. The hydraulic model includes a piston pump that generates the waves. Our result indicates that wave propagation and reflection can create pumping mechanism in a compliant aorta. Similar to an impedance pump, the net flow and the flow direction depends on the frequency of the waves, compliance of the aorta, and the piston stroke.

12:14PM H16.00009 Characterization of human left ventricle flow patterns using ultrasound and Lagrangian coherent structures , SAHAR HENDABADI, Illinois Institute of Technology, JUAN CARLOS DEL ALAMO, University of California, San Diego, YOLANDA BENITO, RAQUEL YOTTI, JAVIER BERMEJO, Department of Cardiology, Hospital General Universitario Gregorio Marañón, Madrid, Spain, SHAWN SHADDEN, Illinois Institute of Technology — We discuss work towards understanding human left ventricle (LV) transport and mixing characteristics in normal subjects and patients with dilated cardiomyopathy. Prior studies have shown that the fluid dynamics in the left ventricle (LV) play a major role in dictating overall cardiac health. This study utilizes a noninvasive method to obtain planar velocity data over the apical long-axis view of the LV from color Doppler and B-mode ultrasound measurements. We use a Lagrangian measure to study unsteady behavior of blood transport inside the LV. We compute finite-time Lyapunov exponent (FTLE) fields to extract Lagrangian coherent structures (LCS) from the empirical data. This application presents a particular challenge to Lagrangian computations due to the presence of moving flux, and no-flux, boundaries. We describe a method for unstructured grid generation from the LV motion, and LCS computation on the deforming unstructured grid. Results demonstrate that LCS reveal the moving boundaries confining the blood volume injected to the LV in diastole and ejected into the aorta in systole. We discuss findings related to the quantification of the LV vortex, whose geometry and motion is thought to be an important indicator of cardiac health.

Monday, November 19, 2012 10:30AM - 12:40PM –

Session H17 Biofluids: Micro-swimming Theory I 28C - Chair: Juan Rodrigo Velez-Cordero, University of California, San Diego

10:30AM H17.00001 Unsteady swimming of small organisms[1] , SHIYAN WANG, AREZOO ARDEKANI, University of Notre Dame — Small planktonic organisms ubiquitously display unsteady or impulsive motion to attack a prey or escape a predator in natural environments. Despite this, the role of unsteady hydrodynamic forces such as history and added mass forces on the low Reynolds number propulsion of small organisms is poorly understood. In this paper, we derive the fundamental equation of motion for an organism swimming by the means of surface distortion in a nonuniform flow at a low Reynolds number regime. We show that the history and added mass forces, that where traditionally neglected in the literature for small swimming organisms, cannot be neglected as the Stokes number increases above unity. For example, these unsteady inertial forces are of the same order as quasi-steady Stokes forces for *Paramecium*. Finally, we quantify the effects of convective inertial forces in the limit of small, but nonzero, Reynolds number regime.

[1]This work is supported by NSF grant CBET-1066545.

10:43AM H17.00002 Advective effects on the propulsion of phoretic micro-swimmers , SEBASTIEN MICHELIN, LadHyX - Ecole polytechnique, DENIS BARTOLO, PMMH - ESPCI — This work focuses on the dynamics of self-propelled spherical particles that can exchange a solute with the surrounding fluid. The propulsion mechanism is based on the interaction of this solute with the surface of the "swimming" particle: concentration gradients along the particle's surface result in a slip velocity distribution, and the particle effectively behaves like an artificial squirmer. In the well-studied diffusive limit, the solute concentration is decoupled from the Stokes flow problem, and the particle's velocity can be directly computed from the distribution of solute flux on the boundary. In this presentation, we identify instead the effect of the solute advection on the propulsive properties of such phoretic micro-swimmers by considering the fully-coupled non-linear problem for the solute concentration and velocity fields around the particle.

10:56AM H17.00003 Achiral rigid magnetically actuated swimmers , HENRY FU, University of Nevada, Reno, U. KEI CHEANG, Drexel University, FARSHAD MESHKATI, University of Nevada, Reno, MINJUN KIM, Drexel University — So far, many magnetically actuated artificial microswimmers have relied on either swimmer flexibility or chiral geometry to overcome constraints on swimming strategies at low Reynolds numbers and achieve propulsion. However, being either flexible or chiral is not a necessary condition for propulsion of microswimmers rotated by external fields. We analyze achiral, rigid swimming using experiment, numerical simulation, and symmetry analysis. Achiral rigid swimming is demonstrated with planar colloidal structures constructed of magnetic beads and rotated by a spatially uniform magnetic field. This swimming is numerically modeled using a boundary element method. Finally, symmetry analysis is used to generically determine which combinations of achiral rigid geometry and magnetic moment can achieve propulsion. For planar colloidal microswimmers the dipole moment must not be perpendicular to a symmetry plane in order to swim.

11:09AM H17.00004 Propulsion in a generalized Newtonian fluid , JUAN RODRIGO VÉLEZ-CORDERO, ERIC LAUGA, University of California at San Diego — The two-dimensional dynamics of an undulating surface has been used as a simplified model to study the transport of fluid by the movement of cilia carpets (so-called envelope model). The collective motion of cilia is idealized as a surface that displaces waves in one direction and whose material points (tips of the cilia) perform a combination of normal and tangential motion with respect to the mean plane. We calculate the mean pumping velocity and rate of work done by an undulating surface in a Generalized Newtonian fluid modeled by the Carreau equation. The influence of the variable viscosity appears only to fourth order in the wave parameter, Ak, where A and k are the wave amplitude and wavenumber respectively. The non-Newtonian effects appear only if both modes of motion, normal plus tangential, are active. The mean rate of work always diminishes for different combinations of normal and tangential motion if the fluid is shear-thinning. Surprisingly, this is not similar for the mean velocity, which for certain motion patterns increases if the fluid is shear-thinning, but for others increases if it is shear-thickening.

11:22AM H17.00005 Diffusion of torqued active particles[1] , MARIO SANDOVAL, ERIC LAUGA, University of California San Diego — Motivated by swimming microorganisms whose trajectories are affected by the presence of an external torque, we calculate the diffusivity of an active particle subject to an external torque and in a fluctuating environment. The analytical results are compared with Brownian dynamics simulations showing excellent agreement between theory and numerical experiments.

[1]This work was funded in part by the Consejo Nacional de Ciencia y Tecnologia of Mexico (Conacyt postdoctoral fellowship to M. S.) and the US National Science Foundation (Grant CBET-0746285 to E.L.).

11:35AM H17.00006 Nematode swimming and turning: locomotion of *C. Elegans* in bulk fluid and thin fluid layers[1] , ALEJANDRO BILBAO, VENKAT PADMANABHAN, Texas Tech University, KENDRA RUMBAUGH, Texas Tech University Health Science Center, SIVA VANAPALLI, JERZY BLAWZDZIEWICZ, Texas Tech University — A millimeter-long nematode *C. Elegans* propels itself by performing sinous undulations, and it turns by assuming strongly curved Ω-shaped body postures. All these stereotyped motions can accurately be described in terms of piecewise-harmonic body curvature, which propagates backwards along the nematode length [PLoS ONE, 7: e40121 (2012)]. We combine our piecewise-harmonic-curvature description with accurate hydrodynamic bead-chain models to investigate swimming efficiency and maneuverability of the nematode in bulk fluid and in a thin fluid layer. We find that the nematode swims faster and maneuvers more efficiently under confinement, because of a larger transverse hydrodynamic resistance. However, the optimal swimming gate is only weakly affected.

[1]This work was supported by NSF grant No. CBET 1059745.

11:48AM H17.00007 Simulation of micro-organisms swimming near ciliated surfaces , HENRY SHUM, ANURAG TRIPATHI, Department of Chemical and Petroleum Engineering, University of Pittsburgh, JULIA YEOMANS, Rudolf Peierls Centre for Theoretical Physics, University of Oxford, ANNA BALAZS, Department of Chemical and Petroleum Engineering, University of Pittsburgh — Ciliated tissues can be found lining the respiratory tract and Fallopian tubes in mammals. The main function of the cilia is to sweep objects such as the ovum, dirt or bacteria in a directed manner. The self-cleaning action of these tissues would be a desirable property for surfaces that are continually submerged and prone to biofouling. We therefore investigate the effect of artificially driven cilia on swimming organisms. In this study we use a 3-D immersed boundary approach, with the fluid flow solved by the lattice Boltzmann method and the immersed objects modeled as elastic structures. Two types of objects are considered: (i) cilia, which are driven by an external field, and (ii) bacteria, which are self-motile and propelled by a rotating helical flagellum. Placing a bacterial cell in the vicinity of a surface covered by an array of actuated cilia yields a rich system to explore. Of particular interest is the possibility of guiding the motion of bacteria towards, along or away from the ciliated surface.

12:01PM H17.00008 Chemotaxis of crawling and swimming *Caenorhabditis Elegans*[1] , AMAR PATEL, ALEJANDRO BILBAO, Texas Tech University, VENKAT PADMANABHAN, Indian Institutes of Technology Kharagpur, ZEINA KHAN, ANDREW ARMSTRONG, Texas Tech University, KENDRA RUMBAUGH, Texas Tech University Health Sciences Center, SIVA VANAPALLI, JERZY BLAWZDZIEWICZ, Texas Tech University — A soil-dwelling nematode *Caenorhabditis Elegans* efficiently navigates through complex environments, responding to chemical signals to find food or avoid danger. According to previous studies, the nematode uses both gradual-turn and run-and-tumble strategies to move in the direction of the increasing concentration of chemical attractants. We show that both these chemotaxis strategies can be described using our kinematic model [PLoS ONE, 7: e40121 (2012)] in which harmonic-curvature modes represent elementary nematode movements. In our chemotaxis model, the statistics of mode changes is governed by the time history of the chemoattractant concentration at the position of the nematode head. We present results for both nematodes crawling without transverse slip and for swimming nematodes.

[1]This work was supported by NSF grant No. CBET 1059745.

12:14PM H17.00009 The Fidelity of Adaptive Phototaxis , IDAN TUVAL, Mediterranean Institute for Advanced Studies, IMEDEA (UIB-CSIC), KNUT DRESCHER, Princeton University, RAYMOND GOLDSTEIN, DAMTP, University of Cambridge — Along the evolutionary path from single cells to multicellular organisms with a central nervous system are species of intermediate complexity that move in ways suggesting high-level coordination, yet have none. Instead, organisms of this type possess many autonomous cells endowed with programs that have evolved to achieve concerted responses to environmental stimuli. Here experiment and theory are used to develop a quantitative understanding of how cells of such organisms coordinate to achieve phototaxis, by using the colonial alga Volvox carteri as a model. It is shown that the surface somatic cells act as individuals but are orchestrated by their relative position in the spherical extracellular matrix and their common photoresponse function to achieve colony-level coordination. Analysis of models that range from the minimal to the biologically faithful shows that, because the flagellar beating displays an adaptive down-regulation in response to light, the colony needs to spin around its swimming direction and that the response kinetics and natural spinning frequency of the colony appear to be mutually tuned to give the maximum photoresponse. These models further predict that the phototactic ability decreases dramatically when the colony does not spin at its natural frequency, a result confirmed by phototaxis assays in which colony rotation was slowed by increasing the fluid viscosity.

12:27PM H17.00010 Granular resistive force theory explains the neuromechanical phase lag during sand-swimming , YANG DING, School of Physics, Georgia Tech, SARAH SHARPE, Interdisciplinary Bioengineering Program, Georgia Tech, DANIEL GOLDMAN, School of Physics, Georgia Tech — Undulatory locomotion is a common gait used by a diversity of animals in a range of environments. This mode of locomotion is characterized by the propagation of a traveling wave of body bending, which propels the animal in the opposite direction of the wave. Previous studies of undulatory locomotion in fluids, on land, and even within sand revealed that the wave of muscle activation progresses faster than the traveling wave of curvature. This leads to an increasing phase lag between activation and curvature at more posterior segments, known as the neuromechanical phase lag. In this study, we compare biological measurements of phase lag during the sand-swimming of the sandfish lizard to predictions of a simple model of undulatory swimming that consists of prescribed kinematics and granular resistive forces. The neuromechanical phase lag measured using electromyography (EMG) quantitatively matches the predicted phase lag between the local body curvature and torque exerted by granular resistive forces. Two effects are responsible for the phase lag in this system: the yaw motion of the body and different integration length over a traveling force pattern for different positions along the body.

Monday, November 19, 2012 10:30AM - 11:48AM –

Session H18 Biofluids: Flapping and Flying II 28D - Chair: Fangjun Shu, New Mexico State University

10:30AM H18.00001 Three-dimensional flow about penguin wings , FLAVIO NOCA, BASSEM SUDKI, MICHEL LAURIA, hepia — Penguins, contrary to airborne birds, do not need to compensate for gravity. Yet, the kinematics of their wings is highly three-dimensional and seems exceedingly complex for plain swimming. Is such kinematics the result of an evolutionary optimization or is it just a forced adaptation of an airborne flying apparatus to underwater swimming? Some answers will be provided based on flow dynamics around robotic penguin wings. Updates will also be presented on the development of a novel robotic arm intended to simulate penguin swimming and enable novel propulsion devices.

10:43AM H18.00002 Numerical simulation of self-propulsion of flapping flexible plates , XI-YUN LU, RU-NAN HUA, University of Science and Technology of China, FLUID MECHANICS TEAM — Self-propulsion of flapping flexible plates has been studied numerically by means of an immersed boundary-lattice Boltzmann method for the fluid dynamics around the plate and a finite element method for the deformable flapping motion of the plate. Both the two- and three-dimensional flexible plates are investigated to reveal the propulsion properties and their differences. As a result of the fluid-plate interaction, three typical movement regimes have been identified and can be briefly described as forward, backward, and irregular movements which mainly depend on the flapping amplitude and bending rigidity. It is found that there exists an optimal range of the bending rigidity for large propulsive speed or high efficiency in the forward movement regime, consistent with the observation and measurement of swimming and flying animals. The results obtained in this study provide physical insights into understanding of the propulsive mechanisms of the flapping fin or wing of animals.

10:56AM H18.00003 Measuring and Analyzing the Birds Flight , ALEXANDER FRIEDL, CHRISTIAN J. KÄHLER, Bundeswehr University Munich — To tackle the long-standing problem of precisely measuring shape and profiling of free-flying birds we developed a technique to determine the shape of naturally textured surfaces. The measurement principle is based on a calibrated stereoscopic camera setup that delivers the height information through the identification of characteristic texture elements in each concurrent camera image using highly developed optical flow algorithms. This allows estimating the motion and height information of each pixel based on the analysis over time. The reconstructed upper surface of the wing is calculated in temporal coherence with the whole image sequence and hence shows low sensitivity to disturbances and high spatial accuracy and resolution. The measurement technique is used to evaluate experimental data obtained within measurement campaigns with two freely flying birds. The slowly, but silently flying barn owl was chosen in contrast to the fast and agile flying lanner falcon. The experiments were carried out within two facilities to respect the different flying performances of the animals and allow for as little disturbances as possible and feasible. Details of the experimental campaigns as well as the measurement methodology will be illustrated during the presentation.

11:09AM H18.00004 Improving Vortex Models via Optimal Control Theory , MAZIAR HEMATI, JEFF ELDREDGE, JASON SPEYER, University of California, Los Angeles — Flapping wing kinematics, common in biological flight, can allow for agile flight maneuvers. On the other hand, we currently lack sufficiently accurate low-order models that enable such agility in man-made micro air vehicles. Low-order point vortex models have had reasonable success in predicting the qualitative behavior of the aerodynamic forces resulting from such maneuvers. However, these models tend to over-predict the force response when compared to experiments and high-fidelity simulations, in part because they neglect small excursions of separation from the wing's edges. In the present study, we formulate a constrained minimization problem which allows us to relax the usual edge regularity conditions in favor of empirical determination of vortex strengths. The optimal vortex strengths are determined by minimizing the error with respect to empirical force data, while the vortex positions are constrained to evolve according to the impulse matching model developed in previous work. We consider a flat plate undergoing various canonical maneuvers. The optimized model leads to force predictions remarkably close to the empirical data. Additionally, we compare the optimized and original models in an effort to distill appropriate edge conditions for unsteady maneuvers.

11:22AM H18.00005 Unsteady lift of a flapping rectangular wing with spanwise stretching-and-retracting , SHIZHAO WANG, GUOWEI HE, Institute of Mechanics, Chinese Academy of Sciences, TIANSHU LIU, Department of Mechanical and Aeronautical Engineering, Western Michigan University, XING ZHANG, Institute of Mechanics, Chinese Academy of Sciences — The unsteady lift acting on a bat-inspired flapping wing model at Reynolds number 300 is numerically investigated. The flapping wing model consists of a rectangular flat-plat with a sinusoidally varying wingspan. The wingspan reaches maximum at the middle of the downstroke, and minimum at the middle of upstroke. It is found that the spanwise stretching-and-retracting during the flapping can enhance lift acting on the wing. The enhancement of lift is not only caused by the difference of lift surface area between the downstroke and upstroke, but also benefits from the increase in the lift coefficient. The enhancement of the vortex force is investigated by examining the flow structures. The spanwise stretching-and-retracting during flapping affects the shedding of the tip vortices and evolution of the leading-edge vortex. The interaction between the detached tip vortices and leading-edge vortex causes a weak negative wake capture mechanisms during the upstroke, which results in a decrease in the magnitude of the minus lift and a increase in the average lift coefficient.

11:35AM H18.00006 Effect of gust on force generation around a robotic hummingbird wing[1] , ELOY MARQUEZ, RUIJUN TIAN, FANGJUN SHU, New Mexico State University — Among the computational, theoretical and experimental studies on the high efficiency flapping flight, many are focused on the mystery of hovering. Most of these studies were conducted under steady in flow conditions. However, real-life ornithopters in the field have to routinely tackle gust and directional changes of the wind. These sudden perturbations could produce significant effect on humming bird hovering due to the small Reynolds numbers. Our experimental work was performed in a water channel using a two degree-of-freedom humming bird model. The dynamic response of the hovering motion to gust from different directions was investigated. PIV was used to measure the effect of the gust on the surrounding flow field including vortex evolution. In addition, a six-component force/torque sensor was used to measure the real-time lift and drag forces generated by the wing with and without gust. Results show that gust changes the magnitude of lift force in one stroke. However, the time-averaged lift force keeps approximately constant.

[1]Supported by Army High Performance Computing Center.

Monday, November 19, 2012 10:30AM - 12:27PM –

Session H19 Multiphase Flows: General I 28E - Chair: Roger Arndt, University of Minnesota

10:30AM H19.00001 ABSTRACT WITHDRAWN –

10:43AM H19.00002 Regarding Multispecies Diffusion and Gradient Driven Transport

, ERIK VOLD, Los Alamos National Lab — A theoretical framework for multispecies fluid transport has been long established in the Maxwell-Stefan equations but interpretations and solution methods appear in the literature to be quite varied. A framework is summarized here for mass drift of species which can include ionized gases with large differences in atomic mass driven by temperature, pressure, and electric potential gradient forces in addition to the usual diffusion driven by concentration gradients. The zero sum over species of mass drift flux closes the $(n_s - -1)$ independent species drift equations for the n_s species and ensures a non-zero molar flux summed over species of different atomic mass. This non-zero species molar flux leads to pressure perturbations, which require a compressible fluids computation to correctly account for the mass average flow and density relaxation. Computations in an initially isothermal binary mixing case illustrate the relaxation of the interfacial density profile by the mass averaged velocity arising from a divergent velocity field. Pressure perturbations associated with boundary reflections and viscosity are shown to have a negligible contribution to the density relaxation compared to the non-zero velocity divergence due to the expansion of each gas during the diffusion. An energy flux, consistent with the species mass diffusion leads to a significant temperature perturbation dominated by a bulk fluid PdV work term rather than the sum over species enthalpy flux. An example for binary diffusion across an interface between species with a large atomic mass difference shows a large asymmetry in species concentration profile unless properly constrained to a net zero sum over species mass flux.

10:56AM H19.00003 Modeling of the Longitudinal Motion of a High Speed Supercavitating Vehicle

, DAVID ESCOBAR, Saint Anthony Falls Laboratory - University of Minnesota, GARY BALAS, University of Minnesota, ROGER ARNDT, Saint Anthony Falls Laboratory - University of Minnesota — High speed supercavitating vehicles offer challenges regarding modeling and control. A mathematical model of the longitudinal motion of a supercavitating test vehicle composed of a cylindrical body, a disk cavitator, and two fins is derived with the aid of experimental data acquired in a high speed water tunnel. The model considers the effect on the vehicle motion of a perturbed flow generated by an oscillating foil gust generator that emulates the effect of flying in the proximity of the sea surface. The vehicle equations of motion and experiments suggest that the fins provide the means for vehicle stability as well as control authority whereas the cavitator only provides control. It was also found that flow oscillations can be modeled as perturbations to the fin angle of attack since variations of the cavitator angle of attack due to the perturbed flow do not contribute to the moments about the vehicle center of gravity. Moreover, an initial view of planing forces generated through large variations of cavitator angle of attack are also presented here. The mathematical description of the vehicle dynamics enables the design of control laws and simulation of the vehicle motion subject to flow perturbations. This research is supported by a grant from the Office of Naval Research.

11:09AM H19.00004 Experimental Validation of Control Systems for a high Speed Supercavitating Vehicle

, ROGER ARNDT, DAVID ESCOBAR, ELLISON KAWAKAMI, Saint Anthony Falls Laboratory - University of Minnesota, GARY BALAS, University of Minnesota — Testing of control systems for a high speed supercavitating vehicle (HSSV) is a challenge in terms of infrastructure and costs. An approach to the control validation of a supercavitating test vehicle is developed. The validation method, referred to as hybrid testing, combines simulation and experimental data in real-time to evaluate the HSSV control systems subject to perturbed flow in the high speed water tunnel at the Saint Anthony Falls Laboratory (Univ. of Minnesota). The test vehicle consists of a cylindrical body, two lateral wedge fins, a pitching disk cavitator used for control, and a ventilation system to insure a fully developed supercavity. A simulation computer uses measurements of the forces applied to the vehicle to compute the vehicle states utilized by the flight computer to control the vehicle simulated motion through the cavitator deflections, which in turn vary the forces applied to the test vehicle. The experimental results validate the suitability of the hybrid test platform, accuracy of the vehicle modeling and control design. as well as the effect of the perturbed flow on the closed-loop system performance. This research is supported by a grant from the Office of Naval Research.

11:22AM H19.00005 Experimental study on gas-liquid bubbly turbulent flow in a large square duct[1]

, HAOMIN SUN, TOMOAKI KUNUGI, Department of Nuclear Engineering, Kyoto University, HIDEO NAKAMURA, Nuclear Safety Research Center, JAEA — Gas-liquid bubbly turbulent flow exists in many industrial areas. Therefore, many experiments for gas-liquid bubbly turbulent flow have been carried out in circular pipes for bubbly turbulent flow model. However, the cross-section of many flow passages are not the circular shape. Since the secondary flow of 2nd kind for single phase turbulent flow in a non-circular duct is well-known, the interaction between the secondary flow of 2nd kind and bubbles in gas-liquid bubbly turbulent flow in the non-circular duct could play an important role. In this study, in order to validate gas-liquid bubbly turbulent flow model in the non-circular duct, measurements were performed in a large square (136mm x 136mm) duct with duct length of 2.8m. The distributions of primary velocity, void fraction and turbulent Reynolds stresses were measured by a hot film probe. It is well-known that the primary velocity distribution of the bubbly flow in a circular pipe has a peak in the pipe center. In contrast, it was found that the primary velocity peaked near the corner of the square duct. In addition, primary velocity distribution changes under various flow conditions were discussed by measuring data of the void fraction and turbulent Reynolds stresses.

[1]Financially Supported by JSPS and G-COE Program(J-051)

11:35AM H19.00006 Circumventing Imprecise Geometric Information and Development of a Unified Modeling Technique for Various Flow Regimes in Capillary Tubes

, BAHMAN ABBASI[1], General Electric Appliances — Owing to their manufacturability and reliability, capillary tubes are the most common expansion devices in household refrigerators. Therefore, investigating flow properties in the capillary tubes is of immense appeal in the said business. The models to predict pressure drop in two-phase internal flows invariably rely upon highly precise geometric information. The manner in which capillary tubes are manufactured makes them highly susceptible to geometric imprecisions, which renders geometry-based models unreliable to the point of obsoleteness. Aware of the issue, manufacturers categorize capillary tubes based on Nitrogen flow rate through them. This categorization method presents an opportunity to substitute geometric details with Nitrogen flow data as the basis for customized models. The simulation tools developed by implementation of this technique have the singular advantage of being applicable across flow regimes. Thus the error-prone process of identifying compatible correlations is eliminated. Equally importantly, compressibility and chocking effects can be incorporated in the same model. The outcome is a standalone correlation that provides accurate predictions, regardless of any particular fluid or flow regime. Thereby, exploratory investigations for capillary tube design and optimization are greatly simplified.

[1]Bahman Abbasi, Ph.D., is Lead Advanced Systems Engineer at General Electric Appliances in Louisville, KY. He conducts research projects across disciplines in the household refrigeration industry.

11:48AM H19.00007 PIV in the two phases of hydrodynamic cavitation in a venturi type section

, SYLVIE FUZIER, SÉBASTIEN COUDERT, OLIVIER COUTIER DELGOSHA, Laboratoire de Mécanique de Lille — The presence of cavitation can affect the performance of turbomachinery. Attached sheet cavities on the blades induce modifications of flow dynamics and turbulence properties. This phenomenon is studied here in a configuration of 2D flow in a venturi type section. Images of the bubbles as well as of the light emitted by fluorescent particles placed in the liquid are recorded simultaneously. Velocities of the bubbles and of the liquid phase are obtained by PIV. The slip velocity is analyzed function of the number of cavitation and other physical parameters. Different levels of turbulence are correlated with different bubble structures in the dipahasic cavity.

12:01PM H19.00008 High-order accurate interface-capturing schemes for gas-liquid flows: pressure and temperature considerations[1] , ERIC JOHNSEN, Mechanical Engineering Department, University of Michigan — Direct simulations of the inertial collapse and rebound of a cavitation bubble are challenging due to the necessity to accurately represent discontinuities (e.g., interfaces and shock waves) and transport processes. A direct application of shock-capturing schemes is known to lead to spurious pressure oscillations and may further generate temperature and conservation errors. The present focus is on high-order accurate schemes (e.g., Weighted Essentially Non-Oscillatory or Discontinuous Galerkin) for interface capturing, which, in analogy to shock capturing, regularize material discontinuities over a few grid points while preserving interfacial conditions. Although approaches have been developed to prevent spurious pressure oscillations in the Euler equations by appropriately coupling a transport equation, temperature spikes may be generated, which in turn lead to transient pressure errors in Navier-Stokes simulations or erroneous mass transfer rates during phase change. These errors are analyzed in the context of gas/liquid and vapor/liquid flows. By appropriately transporting the relevant parameters of a stiffened equation of state, pressure, temperature and conservation errors can be prevented. Results pertaining to cavitation-bubble collapse will be presented.

[1]This work was supported by ONR grant N00014-12-1-0751.

12:14PM H19.00009 Direct numerical simulation of turbulent supercritical flow and heat transfer of water in a vertical pipe[1] , JUNG YUL YOO, SANG HOON LEE, Seoul National University, JOONG HUN BAE, Koreanair R&D Center — Turbulent heat transfer to supercritical-pressure water flowing in a heated vertical tube is investigated using direct numerical simulation. A conservative space-time discretization scheme for variable-density flows at low Mach numbers is adopted to treat steep variations of fluid properties at supercritical pressure just above the thermodynamic critical point, where the fluid properties at such conditions are obtained using PROPATH and used in the form of tables. The buoyancy influence induced by strong variation of density across the pseudo-critical temperature proves to play an important role in turbulent flow and heat transfer at supercritical state. The predicted wall temperature shows localized peaks in the axial distribution. Localized heat transfer impairment of the supercritical-pressure water is found to occur where turbulent energy diffusion is locally suppressed due to the influence of buoyancy. Although the present DNS has been performed at a much lower Reynolds number than that of typical experimental conditions, the peculiar characteristics of supercritical heat transfer including both enhancement and local deterioration are well predicted, in particular, the occurrence of double hot spots.

[1]The support of Priority Research Centers Program through the National Research Foundation funded by the Ministry of Education, Science and Technology (2011-0029613), Republic of Korea is gratefully acknowledged.

Monday, November 19, 2012 10:30AM - 12:27PM –

Session H20 Turbulent Boundary Layers V: Experiments 30A - Chair: Karen Flack, United States Naval Academy

10:30AM H20.00001 Skin-friction and Reynolds number scaling of turbulent channel flow[1] , MICHAEL SCHULTZ, KAREN FLACK, United States Naval Academy — An experimental study was conducted on smooth-wall, fully-developed, turbulent channel flow. The Reynolds number (Re_m) based on the channel height and the bulk mean velocity ranged from 10,000 – 300,000. Measurements of the flow rate and the streamwise pressure gradient allowed the skin-friction coefficient (C_f) to be determined, and its variation with Reynolds number will be discussed and compared with previous investigations. Two-component LDV measurements were also made at friction Reynolds numbers $Re_\tau = 1{,}000 - 6{,}000$. The scaling of both the mean flow and the Reynolds stresses will be examined. In particular, the variation in these quantities with Reynolds number will be discussed.

[1]This research was funded by ONR.

10:43AM H20.00002 Measurements of the wall-normal velocity component in very high Reynolds number pipe flow[1] , MARGIT VALLIKIVI, MARCUS HULTMARK, ALEXANDER J. SMITS, Princeton University — Nano-Scale Thermal Anemometry Probes (NSTAPs) have recently been developed and used to study the scaling of the streamwise component of turbulence in pipe flow over a very large range of Reynolds numbers. This probe has an order of magnitude higher spatial and temporal resolution than regular hot wires, allowing it to resolve small scale motions at very high Reynolds numbers. Here use a single inclined NSTAP probe to study the scaling of the wall normal component of velocity fluctuations in the same flow. These new probes are calibrated using a method that is based on the use of the linear stress region of a fully developed pipe flow. Results on the behavior of the wall-normal component of velocity for Reynolds numbers up to 2 million are reported.

[1]Supported under NR Grant N00014-09-1-0263 (program manager Ron Joslin) and NSF Grant CBET-1064257 (program manager Henning Winter).

10:56AM H20.00003 Experimental study of the boundary layer properties in ultimate Taylor-Couette flow , SANDER HUISMAN, ROELAND VAN DER VEEN, CHAO SUN, DETLEF LOHSE, Physics of Fluids group, University of Twente, Netherlands — We report high-resolution measurements of the properties of the velocity boundary layer in turbulent Taylor-Couette flow using time-resolved particle image velocimetry (PIV). The experiments are performed in the Twente Turbulent Taylor-Couette facility (T3C). The Taylor number is varied from 10^8 to 10^{13}, which covers the ultimate turbulence regime and the transition regime. We also change the rotation ratio of the inner and outer cylinders. The boundary layer profile, thickness, and scaling behavior are experimentally examined. In addition, the measured results are closely compared to the boundary layer properties in turbulent Rayleigh-Bénard flow.

11:09AM H20.00004 New insight on flow development and two-dimensionality of turbulent channel flows , H. NAGIB, R. VINUESA, E. BARTRONS, M. MUÑOZ, G. SUBASHKI, IIT, Chicago, Y. SUZUKI, Nihon University, Japan — The experimental conditions required for a turbulent channel flow to be considered fully-developed and nominally two-dimensional remain a challenging objective. Oil film interferometry (OFI) and static pressure measurements were carried out over the range $200 < Re_\tau < 800$ in an adjustable-geometry channel flow facility. Three-dimensional effects were studied by considering different aspect ratio (AR) configurations, and also by fixing the AR and modifying the hydraulic diameter D_H of the section. The conditions at the centerplane of the channel were characterized through the local skin friction from the OFI and the centerline velocity at three different streamwise locations, as well as the wall shear based on the streamwise global pressure gradient. The skin friction obtained from the pressure gradient overestimated the local shear measurements obtained from the OFI, and did not reproduce the same AR-dependence observed with OFI. Differences between the local and global techniques were also reflected in the flow development. Development length of high-aspect-ratio channels scales with the hydraulic diameter of the section, and is around 200 channel full-heights H, much larger than the values of around $100 - 150$ H previously reported in the literature.

11:22AM H20.00005 Multi-component measurements in high Reynolds number turbulent boundary layers , RIO BAIDYA, JIMMY PHILIP, NICHOLAS HUTCHINS, JASON MONTY, IVAN MARUSIC, The University of Melbourne — Measurements, with highly resolved spectra are obtained in high Reynolds number (Re_τ) turbulent boundary layers using sub-miniature cross-wires. The probe consists of 2.5μm diameter platinum wires welded across the sharpened stainless steel prong tips, contained in a volume of $0.4 \times 0.4 \times 0.2$mm^3. Velocity profiles are measured at various stream-wise positions with nominally matched unit Reynolds number (U_∞/ν). In this manner the same probe geometry affords approximately matched viscous-scaled sensor length (l^+) and sensor spacing (Δs^+) across the entire range of Re_τ, such that Reynolds number trends can be observed free of spatial resolution effects. The probes have matched measurement volumes of approximately $14 \times 14 \times 7$ ($\pm 10\%$) viscous length scales across all Re_τ. The prong is inclined at 10° to the horizontal, permitting measurements close to the wall while also minimising blockage effects. The prongs are fabricated to account for this inclination, ensuring that the sensing elements remain parallel to the wall at the desired prong orientation. The resulting highly resolved multi-component velocity statistics up to $Re_\tau = 10,000$ and their associated trends against Re_τ will be presented.

11:35AM H20.00006 Turbulent convection velocities in a turbulent boundary layer , ROELAND DE KAT, BHARATHRAM GANAPATHISUBRAMANI, University of Southampton — Turbulent convection velocities in a turbulent boundary layer are of crucial importance for bringing together theory and experiment. In this study, we determine probability density functions (pdfs) of turbulent convection velocities per wavenumber k_x –using a phase-spectral approach– from time-resolved PIV measurements in a stream-wise wall-normal plane of a turbulent boundary layer at $Re_\tau \approx 2700$. A field-of-view covering approximately $2 \times 0.5\delta$ with high spatial, $l^+ = 20$, and temporal resolution, $\Delta t^+ = 0.7$, allows us to determine convection velocities for a range of different wall-normal locations ($y/\delta : 0.02 - 0.47$; $y^+ : 60 - 1260$). The results indicate that the mode of the pdfs coincides with the local mean velocity and that there is a considerable spread around this mode. In the talk, a detailed description of turbulent convection velocities with wall-normal distance will be presented.

11:48AM H20.00007 Correction of Pressure Data Close to the Wall in Turbulent Boundary Layer , YOSHIYUKI TSUJI, Nagoya University, YOSHINOBU YAMAMOTO, Yamanashi University — We have developed a small pressure probe and measured both static pressure and wall pressure simultaneously in turbulent boundary layers up to Reynolds numbers based on the momentum thickness 44000. Experimental data were obtained by the same person using the same techniques at the three large facilities. We find that the measured pressure data are contaminated by the artificial background noise induced by test section and are also affected by the flow boundary conditions. By analyzing data from different wind tunnels acquired at the same Reynolds number, we evaluate the effect of background noises and boundary conditions on the pressure statistics. We also compare the experimental results with results of direct numerical simulations and discuss differences in boundary conditions between real and simulated wind tunnels. The results have already been reported in TSFP7. In the present paper, we are interested in the interaction with pressure probe and the wall. It is very difficult to measure the pressure fluctuation close to the wall. We discuss the effect of solid wall on the pressure data and suggest the method how to correct the data. The measured data are compared with those of DNS.

12:01PM H20.00008 Turbulent boundary layer investigation at large Re with micron resolution , CHRISTIAN J. KAEHLER, CHRISTIAN CIERPKA, SVEN SCHARNOWSKI, Institute of Fluid Mechanics and Aerodynamics, Bundeswehr University Munich — The reliable measurement of statistical quantities in turbulent boundary layer flows down to the wall is a challenging problem for many decades. However, due to the progress in laser based experimental techniques in the last years, it is now non-intrusively possible to measure statistical quantities, such as the mean velocity profile, wall-shear stress, Reynolds stresses or the probability density functions of the turbulent fluctuations, with micron resolution (Kähler et al. Exp. Fluids, 2012). The high spatial resolution allows for accurate measurements as typical bias errors, caused by spatial averaging effects of the probe size, can be avoided. Using advanced optical techniques, we have investigated a turbulent boundary layer flow along a 22 m long flat plate, installed in a wind-tunnel with a 2m by 2m cross-section, at different Reynolds numbers. The statistical results of the investigation will be discussed in the contribution.

12:14PM H20.00009 Canonical boundary layer properties at high Reynolds number as measured in the UNH Flow Physics Facility , PASCHAL VINCENTI, CALEB MORRILL-WINTER, JOSEPH KLEWICKI, CHRISTOPHER WHITE, MARTIN WOSNIK, University of New Hampshire — This presentation describes the characteristics of the flow within the Flow Physics Facility (FPF) at the University of New Hampshire. Having a test section length of 72m, the FPF employs the "big and slow" solution to obtaining well-resolved turbulent boundary layer measurements at high Reynolds number. We report on experiments that investigate the wind speed and Reynolds number capability, spanwise uniformity, streamwise pressure gradient, and free-stream turbulence intensity in the FPF. Single element hot-wire measurements of the boundary layer statistical profiles (up to fourth central moment) are presented. These experiments used standard 1mm sensors to generate spatially and temporally well-resolved measurements over the Karman number range 2000 - 20000. Integral parameters and spectra, at a variety of stream-wise locations and Reynolds numbers, are presented, and compared to existing data. For a wide range of test conditions, the FPF is shown to provide high-resolution access to the turbulence of the canonical boundary layer at high Reynolds number.

Monday, November 19, 2012 10:30AM - 12:27PM –

Session H21 Turbulence Simulation: Sensitivity/Uncertainty 30B - Chair: Qiqi Wang, Massachusetts Institute of Technology

10:30AM H21.00001 Stabilized sensitivity analysis of scalar mixing in laminar and turbulent jet in crossflow , RUI CHEN, QIQI WANG, PATRICK BLONIGAN, MIT Department of Aeronautics and Astronautics ACDL — Solutions to both linearized Navier-Stokes equation and its adjoint equation grow exponentially in high Reynolds number turbulent flows, making sensitivity analysis of long time averaged, statistical quantities difficult. This talk presents a stabilization scheme by adding numerical viscosity to stabilize these equations. This adjoint stabilization scheme is applied to scalar mixing in laminar and turbulent jets in cross flow. We vary two parameters, the Reynolds number Re and the jet-to-crossflow velocity ratio R. At low Re and low R, steady flow field and adjoint solution is obtained. At medium Re and low R, the flow is unsteady and the adjoint solution grows; we analyze the effect of additional stabilizing viscosity in the linearized and adjoint equations to the accuracy of computed sensitivity. The effect of numerical viscosity at high Re and large R, when the flow field becomes turbulent and chaotic, is also analyzed. By comparing these cases, we summarize the performance of the stabilization scheme.

10:43AM H21.00002 Impact of numerical errors on the turbulent mixing of high Schmidt number passive scalars , YUAN XUAN, SIDDHARTHA VERMA, GUILLAUME BLANQUART, California Institute of Technology — Numerical errors associated with scalar transport schemes can affect significantly the mixing of high Schmidt number passive scalars. In this work, we present an analysis of the impact of these errors on the scalar transport characteristics in homogeneous isotropic turbulence and turbulent mixing layers. These two configurations are selected as representatives of different regions of a reacting turbulent jet. We evaluate scalar energy and dissipation spectra, as well as the probability density functions of the scalar and its dissipation rate. This analysis is performed at various grid resolutions, using several different Eulerian and Semi-Lagrangian transport schemes. The results are used to establish the accuracy of these schemes in capturing and preserving the small-scale turbulent structures. It is shown that Eulerian schemes require comparatively higher grid resolution to produce results independent of the mesh size. Conversely, semi-Lagrangian schemes are capable of achieving comparable accuracy at lower grid resolutions, resulting in significant reductions in computational cost. We use the results to propose grid resolution criteria to ensure scheme independent results for high Schmidt number scalar transport in homogeneous isotropic turbulence and turbulent mixing layers.

10:56AM H21.00003 Adjoint Sensitivity Computation for Unsteady, Periodic Fluid Flows , STEVEN GOMEZ, QIQI WANG, Massachusetts Institute of Technology — Adjoint sensitivity analysis is an important computational method to assist in engineering design and optimization problems, and can be used to efficiently compute the sensitivity of an objective function with respect to many parameters simultaneously. While these methods are popular for steady problems, there are issues when extending them to periodic and chaotic systems. Some techniques, such as windowing, have made progress at computing time average sensitivities for periodic systems; however, they do not provide a time accurate representation of the desired sensitivity. We propose a new method of adjoint computation for periodic systems that produces a time accurate sensitivity by computing and correcting two adjoint systems simultaneously. By decomposing input perturbations into components that produce pure phase shifts and no phase shifts, we derive the governing equations for the time accurate adjoint. We then propose an algorithm for computing this adjoint solution with the added overhead of storing and computing one additional adjoint variable. This algorithm is tested on the Van der Pol oscillator and a CFD simulation of vortex shedding behind a cylinder. Possible extensions to chaotic systems, such as turbulent fluid flows, will also be examined.

11:09AM H21.00004 Estimating Uncertainties in Statistics Computed from DNS[1] , NICHOLAS MALAYA, RHYS ULERICH, TODD OLIVER, ROBERT MOSER, University of Texas at Austin — Direct numerical simulation (DNS) of turbulence is a critical tool for investigating the physics of turbulent flows and for informing and developing engineering turbulence models. For instance, flow statistics obtained from DNS are commonly used as "truth data" for the calibration and evaluation of turbulence models. Thus, like experimental data, uncertainty estimates are a necessary component of the reported output. In DNS, uncertainties in the computed statistics arise from two sources: finite sampling and the discretization of the Navier-Stokes equations. Here, we apply estimators for both sources of error. Finite sampling errors are estimated using the "effective sample size," which accounts for the fact that the instantaneous data are correlated. Discretization errors are estimated using data from simulations with varying time step and mesh spacing. The performance of these estimators is tested for several statistics using DNS of turbulent channel flow at low Reynolds number ($Re_\tau \approx 180$).

[1]This work is supported by the Department of Energy National Nuclear Security Administration under Award Number [DE-FC52-08NA28615].

11:22AM H21.00005 Representing Turbulence Model Uncertainty with Stochastic PDEs , TODD OLIVER, ROBERT MOSER, The University of Texas at Austin — Validation of and uncertainty quantification for extrapolative predictions of RANS turbulence models are necessary to ensure that the models are not used outside of their domain of applicability and to properly inform decisions based on such predictions. In previous work, we have developed and calibrated statistical models for these purposes, but it has been found that incorporating all the knowledge of a domain expert—e.g., realizability, spatial smoothness, and known scalings—in such models is difficult. Here, we explore the use of stochastic PDEs for this purpose. The goal of this formulation is to pose the uncertainty model in a setting where it is easier for physical modelers to express what is known. To explore the approach, multiple stochastic models describing the error in the Reynolds stress are coupled with multiple deterministic turbulence models to make uncertain predictions of channel flow. These predictions are compared with DNS data to assess their credibility. This work is supported by the Department of Energy [National Nuclear Security Administration] under Award Number [DE-FC52-08NA28615].

11:35AM H21.00006 Breakdown of Sensitivity Analysis in Chaotic, Turbulent Fluid Flows[1] , QIQI WANG, PATRICK BLONIGAN, MIT, JUNHUI GAO, BUAA — Sensitivity analysis is a class of algorithms for calculating derivatives of output quantities with respect to input parameters in computational fluid dynamics simulations. It is an essential ingredient for data assimilation, aerodynamic design, uncertainty quantification and flow control. Sensitivity analysis in high fidelity simulations of turbulent flows (DNS or LES) is challenging. These simulations are true to the chaotic nature of turbulence: instantaneous flow fields are very sensitive to perturbations in parameters and geometry; while long time averaged, statistical quantities are often well behaved functions of parameters and geometry. Consequently, sensitivity of statistics cannot be computed by taking the statistics of sensitivity. For example, averaging the sensitivity of instantaneous drag over a long time do not give the sensitivity of the mean drag. This talk first discuss the mathematical reason of the breakdown of sensitivity analysis of statistical quantities in chaotic systems. We then demonstrate the breakdown on a number of examples, including airfoil at high angle of attack, a cylinder in crossflow, and a turbulent jet in a cross flow. We show that positive Lyapunov exponents in these systems lead to divergence of the conventional sensitivity analysis.

[1]The authors gratefully acknowledge support from a subcontract of DOE PSAAP at Stanford, and the Center for Turbulence Research summer program

11:48AM H21.00007 New Methods for Sensitivity Analysis in Chaotic, Turbulent Fluid Flows , PATRICK BLONIGAN, QIQI WANG, Massachusetts Institute of Technology — Computational methods for sensitivity analysis are invaluable tools for fluid mechanics 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 flowfields, such as those obtained using high-fidelity turbulence simulations. Also, a number of dynamical properties of chaotic fluid flows, most notably the "Butterfly Effect," make the formulation of new sensitivity analysis methods difficult. This talk will outline two chaotic sensitivity analysis methods. The first method, the Fokker-Planck adjoint method, forms a probability density function on the strange attractor associated with the system and uses its adjoint to find gradients. The second method, the Least Squares Sensitivity method, finds some "shadow trajectory" in phase space for which perturbations do not grow exponentially. This method is formulated as a quadratic programing problem with linear constraints. This talk is concluded with demonstrations of these new methods on some example problems, including the Lorenz attractor and flow around an airfoil at a high angle of attack.

12:01PM H21.00008 Quantification of epistemic uncertainties in RANS turbulence models , MARIA VITTORIA SALVETTI, LUCA MARGHERI, University of Pisa - Italy, MARCELLO MELDI, PIERRE SAGAUT, Pierre et Marie Curie University - Paris 6 - France — Thanks to its limited computational requirements, the RANS approach has extensively been used and is still used to predict the low-order statistics of high Reynolds number turbulent flows. The main drawback is that an universal setup of the closure turbulence models has proved to be elusive. The free parameters present in turbulence models are usually derived from estimated deterministic values of some properties of benchmark turbulent flows, as e.g. the energy power law exponent for decaying homogeneous isotropic turbulence or the value of the von Karman constant. The free parameters present in different well-known turbulence models are obtained herein by considering the underlying properties as random variables over a bounded range. This range has been recovered from the results reported in literature for the relevant properties, so that the considered epistemic uncertainty is realistic. The sensitivity to this uncertainty of the results of turbulent channel flow RANS simulations is then investigated for different Reynolds numbers. The solution over the continuous multi-dimensional uncertainty space of the considered random variables is reconstructed through the application of a surrogate model (response surface) obtained by means of generalized Polynomial Chaos.

12:14PM H21.00009 Evaluation and Quantification of Uncertainty of RANS turbulence and turbulent mixing models for a separated flow[1] , CATHERINE GORLE, Stanford University, RICCARDO ROSSI, Universita di Bologna, Stanford University, GIANLUCA IACCARINO, Stanford University — The inability of the k-ε and k-ω RANS turbulence models to correctly predict flow separation and reattachment limits the reliability of simulations of complex flows. When also predicting the turbulent diffusion of a scalar, algebraic models for the scalar fluxes, which rely on the turbulent viscosity or Reynolds stresses predicted by the turbulence model, introduce additional errors in the solution. In the present work these errors are evaluated by comparing the Reynolds stresses obtained from the RANS models to DNS results for the flow over a wavy wall. The effect of the coupling to the mean flow is eliminated by freezing the flow to the time-averaged DNS flow field and only solving the transport equations for the turbulence quantities. The goal of this analysis is to establish a statistical model for the errors in the modeled Reynolds stresses. The errors are investigated in terms of the turbulence kinetic energy and eigenvalues and eigenvectors of the anistropy tensor. The statistical models for these quantities are used to perturb the Reynolds stresses and quantify the uncertainty in the location of the reattachment point. By also introducing the perturbations in an algebraic model formulation for the scalar fluxes the capability of quantifying the uncertainty in scalar mixing is investigated.

[1]This material is based upon work supported by the DoE [NNSA] under Award Number NA28614.

Monday, November 19, 2012 10:30AM - 11:48AM –

Session H22 Turbulence Mixing II: Heat Transport 30C - Chair: Stefano Leonardi, University of Puerto Rico

10:30AM H22.00001 Direct Numerical Simulations (DNS) of the thermal field in a turbulent channel flow with spanwise sinusoidal blowing/suction , CAN LIU, GUILLERMO ARAYA, LUCIANO CASTILLO, Department of Mechanical Engineering, Texas Tech University, STEFANO LEONARDI, Mechanical Engineering Department, University of Puerto Rico at Mayagüez — Direct Numerical Simulations (DNS) of an incompressible turbulent channel flow with given local perturbations at the walls are performed. Steady blowing and suction are applied at both walls by means of five spanwise holes. The sinusoidal perturbing velocity is considered at several amplitudes (0.025, 0.1 and 0.2 based on the centerline velocity) as well as at two different angles (30 and 40 degrees with respect to the flow direction) in order to explore its effects on the adiabatic efficiency. The Reynolds number of the unperturbed case is Re = 394 and the molecular Prandtl number is Pr = 0.71. Isoflux conditions are assumed for the lower and the upper walls. Furthermore, turbulence statistics, energy budgets and energy spectra are going to be examined for the velocity and thermal fields.

10:43AM H22.00002 ABSTRACT WITHDRAWN –

10:56AM H22.00003 Devising scaling parameters for wall bounded turbulent thermal transport[1] , CHIRANTH SRINIVASAN, DIMITRIOS PAPAVASSILIOU, University of Oklahoma — Scaling of turbulent heat transfer from a wall with the friction temperature does not work well for all turbulence quantities and for all Prandtl number (Pr) fluids. The lack of a comprehensive database covering heat transfer statistics for a wide range of Pr and Reynolds numbers (Re) has hindered recent investigations to obtain more appropriate scaling parameters. This study uses turbulent transport statistics from our extensive database to propose a new scaling framework for turbulent transport. The database is obtained by using DNS in conjunction with Lagrangian tracking of heat markers to generate heat transfer statistics in a turbulent channel flow. The simulated cases involve applying uniform heat flux on one wall while maintaining the other wall adiabatic, or applying uniform heat flux on both walls. The channel half-height is equal to 150 and 300 in viscous wall units and the Pr varies between 0.1 and 50,000. It is found that the peak value of the turbulent heat flux appears to be a parameter that can be used to asymptotically collapse the mean and the fluctuating scalar profiles to corresponding single profiles for different Re and Pr, thus, establishing the peak normal turbulent heat flux as an important scaling parameter.

[1]NSF CBET-0651180 and XSEDE TG-CTS090025

11:09AM H22.00004 Preliminary Results in an Ablation Wind Tunnel[1] , MICHAEL ALLARD, CHRISTOPHER WHITE, University of New Hampshire, YVES DUBIEF, University of Vermont — We have constructed a small-scale boundary layer wind tunnel to study rapid turbulent ablation of a surface subjected to a heated flow to better understand the complex coupling between an erodible surface and an eroding agent. The test section of the tunnel is 303mm $\times$ 141mm cross-section and 2.75m in length. The turbulence management section contains a 4:1 contraction, honeycomb, and 4 screens of decreasing mesh size. A 3/4HP fan is used to establish air flow in the tunnel and a frequency controller is used to control and maintain constant flow speed, which can be varied between 0.1 to 18m/s. A bank of electric heaters and a proportional integral derivative (PID) controller is used to maintain constant inlet air temperature. Characterization of the flow field over a non-ablative surface is reported, and qualitative observations of vortex induced erosion patterns on an ablative wall are presented.

[1]This work is supported by the NSF (CBET-0967224)

11:22AM H22.00005 A POD analysis of rough surface pressure and temperature fluctuations in a spatially developing turbulent boundary layer , LUCIANO CASTILLO, JENSEN NEWMAN, Texas Tech University, RONALD ADRIAN, Arizona State University, YI CHEN, Rensselaer Polytechnic Institute — It is desired to gain a better understanding of how the presence of surface roughness in a turbulent boundary layer affects phenomena such as heat transfer and aeroacoustic noise generation. With this in mind, it is expected that by examining the most dominant features of these scalar fields in flows with and without surface roughness progress could be made towards this end. Hence, a Proper Orthogonal Decomposition (POD) was performed on the surface temperature and pressure fluctuations of a DNS simulation with a smooth wall, and one with a rough wall. The rough topography corresponded to an actual rough topography measured from 24 grit sand paper. Reynolds numbers based on momentum thickness ranges for the simulations were 1922-2269 for the smooth and 2077-2439 for the rough. The low order POD modes correspond to the dominant features of interest in these scalar fields. By comparing the smooth and rough cases, important qualitative differences in the dominant features were observed.

11:35AM H22.00006 Effect of surface heating on the drag crisis of sphere , MASAYA MUTO, HIROAKI WATANABE, Central Research Institute of Electric Power Industry, MAKOTO TSUBOKURA, Hokkaido University — The characteristics of flow past a heated sphere are investigated at around critical Reynolds number in conditions using three-dimensional numerical simulation in which temperature dependence of fluid properties such as density and viscosity is exactly considered. Boussinesq approximation is no longer applicable due to large temperature difference adopted in this study. And the order of the buoyancy effect becomes relatively small compared to inertia effect in present Reynolds number region. The result shows that drag coefficient of the heated sphere in drag crisis region becomes larger than that of the unheated case and it increases up to the coefficient found in subcritical region. This is because the temperature difference between the sphere and ambient fluid strongly affects the flow separation points, resulting in small recovery of the pressure in the wake and reduction of the temporal fluctuation of the lift force acting on the sphere. These effects are considered to attribute to the temperature dependence of fluid properties in the vicinity of the sphere and effect on the transient of the boundary surface.

Monday, November 19, 2012 10:30AM - 12:40PM –

Session H23 Turbulent Boundary Layers VI: Compressible 30D - Chair: Krishnan Mahesh, University of Minnesota

10:30AM H23.00001 Measurement in a Hypersonic Turbulent Boundary Layer Using PIV , OWEN WILLIAMS, ALEXANDER SMITS, Princeton University — Experiments are reported on measuring turbulence in a flat plate boundary layer at Mach 7.4 using planar PIV in order to examine Morkovin's hypothesis and scaling at Mach numbers greater than 5. PIV measurements in hypersonic flow are hampered by high dynamic range requirements and low flow density, which leads to stringent particle sizing requirements to avoid particle lag. In addition, high shear can lead to a bias in many cross-correlation algorithms. Experiments to determine the frequency response of a range of titanium dioxide particles using the response across a shock will be detailed. Additionally, the conditions for the appropriate initial conditions for boundary layer development, such as the selection of size and type of tripping device and appropriate development length for the establishment of a fully turbulent boundary layer will be examined.

10:43AM H23.00002 Predictive Inner-Outer Model for Turbulent Boundary Layers Applied to Supersonic DNS , PINO MARTIN, CLARA HELM, University of Maryland, College Park — A predictive wall model for subsonic turbulent boundary layers is modified for application to supersonic and hypersonic boundary layers. The original model is based on an observed modulation of the turbulence in the inner layer by the large scale motions in the logarithmic layer. Evidence that this modulating effect also exists in compressible turbulent boundary layers is presented. The appropriate scalings applied to the model to deal with compressibility effects is discussed. Spectrally resolved data sets from direct numerical simulation (DNS) of Mach 3 and Mach 7 turbulent boundary layers are used for validation of the modified model. Predicted inner layer velocity fields at the same flow conditions as the DNS data are constructed and a comparison of statistics, such as spectra and moments, of the predicted and DNS flow field parameters is presented. This work is supported by the Air Force Office of Scientific Research under grant AF/9550-10-1-0164.

10:56AM H23.00003 Direct Numerical Simulation of a Mach 2.25 Turbulent Boundary Layer Over A Compliant Surface[1] , CHRISTOPHER OSTOICH, DANIEL BODONY, PHILIPPE GEUBELLE, University of Illinois at Urbana-Champaign — Future high-speed air vehicles will be lightweight, flexible, and reusable, and thus susceptible to dynamic coupling between the boundary layer and vehicle surface. Current analysis techniques of boundary layer properties based on rigid surface assumptions may be invalid in this flight regime, and experimental approaches are difficult and expensive. Analyses involving accurate numerical predictions are therefore needed to provide data in this high-speed, coupled environment. Results of a Mach 2.25 turbulent boundary layer evolving over a thin, compliant panel are investigated using direct numerical simulation in the fluid coupled to a geometrically non-linear, thermomechanical finite element solver for the solid. The coupled response of the boundary layer-panel system are presented and compared to the rigid panel case. Panel deformation and fluid flow modification due to the presence of the compliant panel are analyzed, with a particular interest in the influence of panel motion on turbulence statistics. The maximum panel deformation extends through the viscous sublayer while space-time data show that the solid supports waves that may be useful for sensing and control.

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

11:09AM H23.00004 DNS of a Mach 3 and Mach 7 Turbulent Boundary Layer: Statistical Description and Scale Decomposed Physics[1] , IZAAK BEEKMAN, Princeton/UMD, PINO MARTIN, University of Maryland — We analyze the statistical properties of supersonic turbulent boundary layers via spatial direct numerical simulations (SDNS). The computational domains are very large with 60 by 10 δ_{inlet} in the streamwise and spanwise directions respectively. The inflow is provided with a rescaling technique, where the recycling station is taken near the outlet, with $Re_\tau \approx 650$. We vary the nominally adiabatic wall boundary condition between a treatment that enforces a null *mean* heat transfer to the wall ($T_w = T_{recovery}, T' = 0$) and an *instantaneously* adiabatic wall with $(\partial T/\partial z)_w = 0$. The data, including spectra, are converged over $600\delta/U_\infty$ and capture the largest scales of the flow. We study broadband relations such as the strong Reynolds analogy (SRA) and Morkovin's scaling of the turbulent shear stresses in scale decomposed manner to gain insight and physical understanding for turbulence modeling.

[1]This work is supported by the University of Maryland Center of Excellence for Testing and Evaluation.

11:22AM H23.00005 Mach-number-invariant mean velocity profile of compressible turbulent boundary layers , YOU-SHENG ZHANG, WEI-TAO BI, ZHEN-SU SHE, College of Engineering, Peking University, FAZLE HUSSAIN, Department of Mechanical Engineering, University of Houston, XIN-LIANG LI, Chinese Academy of Sciences — A series of Mach-number- (M) invariant scalings are derived for compressible turbulent boundary layers (CTBLs), leading to a viscosity weighted transformation for the mean velocity profile (MVP) that is superior to van Driest transformation. The new scalings are derived from an analysis of turbulent kinetic energy balance equation, and their finding substantiates Morkovin's hypothesis. In particular, a boundary layer edge, δ_{vw}, is defined by equaling the intensities of wall-normal and spanwise velocity fluctuations, and is shown to better represent the M-invariant structure of CTBLs. The M-invariant mixing length has a weight of $\sqrt{\bar{\rho}^+}\bar{\mu}^+$ to the Prandtl's mixing length, which leads to a viscosity weighted transformation for the MVP of CTBLs, in contrast to the density weighted van Driest transformation. The theory is validated by direct numerical simulation of spatially-developing adiabatic CTBLs with M up to 6. These results suggest new tools for validating turbulence models and improving computations in CTBLs.

11:35AM H23.00006 Direct numerical simulation of a hypersonic shock wave/turbulent boundary layer interaction[1] , STEPHAN PRIEBE, PINO MARTIN, University of Maryland — The direct numerical simulation of a hypersonic shock wave/turbulent boundary layer interaction generated by a 33-degree compression ramp is presented. The fully-turbulent inflow boundary layer is at Mach 7.2, and the Reynolds number based on momentum thickness is $Re_\theta = 3500$. The evolution of the mean and fluctuating field through the interaction region and the properties of the low-frequency unsteadiness are investigated.

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

11:48AM H23.00007 Wall-modeled large-eddy simulations of shock/turbulent-boundary layer interaction in a duct[1] , IVAN BERMEJO-MORENO, JOHAN LARSSON, LAURA CAMPO, JULIEN BODART, DAVID HELMER, FRANK HAM, JOHN K. EATON, Stanford University — We present wall-modeled LES of the interaction of an oblique shock wave and a turbulent boundary layer in a low-aspect-ratio duct, following the experiment by Campo, Helmer & Eaton, 2012. A $M = 2.05$ air stream is deflected by a small compression wedge that spans the top wall of the duct at 20°, generating an oblique shock that reflects off the turbulent boundary layer at the bottom wall. The Reynolds number of the incoming flow based on the boundary layer momentum thickness is $Re_\theta = 6,500$. Simulations are performed with a control-volume-based, finite-volume solver of the filtered, compressible Navier-Stokes equations, utilizing a grid-based blend of non-dissipative central and dissipative upwind fluxes, Vreman (2004) subgrid scale model and ENO shock-capturing scheme, active only near shocks. An equilibrium wall model (Kawai and Larsson, 2012) is used. Turbulence is generated at the inflow from experimental data. Simulation results are compared to PIV data on four planes normal to the spanwise coordinate. A grid-convergence study is performed. Two heights of the compression wedge are considered, resulting in an increased strength of the interaction. Simulation results are also used to study the evolution of corner flows, complementing experimental findings.

[1]The authors acknowledge financial support provided by the USDoE under the Predictive Science Academic Alliance Program (PSAAP) at Stanford University.

12:01PM H23.00008 Direct Numerical Simulations of Turbulent Boundary Layers Over A Circular Aperture , QI ZHANG, DANIEL BODONY, University of Illinois at Urbana Champaign — Motivated by the use of acoustic liners to reduce jet engine and aircraft noise, we use direct numerical simulation to study the interaction of a turbulent Mach 0.5 boundary layer with a circular aperture connected to a honeycomb cavity under acoustic excitation. The geometry and flow conditions correspond to experiments conducted at NASA Langley. The hole, whose diameter is on the order of the boundary layer's momentum thickness, interacts with the boundary layer in qualitatively different ways depending on the acoustic forcing amplitude. The influence of the hole on the boundary layer is quantified under a range of acoustic excitations and the details of the hole/boundary layer interaction will be presented. The acoustic impedance of the hole is determined, compared to experimentally educed values, and related to the dynamics of the hole/boundary layer interaction. These analyses will be helpful for improved understanding and low-order models of aircraft acoustic liners at realistic operating conditions.

12:14PM H23.00009 Large Eddy Simulation of a Film Cooling Technique with a Plenum , SURANGA DHARMARATHNE, NARENDRAN SRIDHAR, GUILLERMO ARAYA, LUCIANO CASTILLO, SIVAPATHASUND PARAMESWARAN, Department of Mechanical Engineering, Texas Tech University — Factors that affect the film cooling performance have been categorized into three main groups: (i)coolant & mainstream conditions, (ii)hole geometry & configuration, and (iii)airfoil geometry Bogard et al.(2006). The present study focuses on the second group of factors, namely, the modeling of coolant hole and the plenum. It is required to simulate correct physics of the problem to achieve more realistic numerical results. In this regard, modeling of cooling jet hole and the plenum chamber is highly important Iourokina et al.(2006). Substitution of artificial boundary conditions instead of correct plenum design would yield unrealistic results Iourokina et al.(2006). This study attempts to model film cooling technique with a plenum using a Large Eddy Simulation.Incompressible coolant jet ejects to the surface of the plate at an angle of 30° where it meets compressible turbulent boundary layer which simulates the turbine inflow conditions. Dynamic multi-scale approach Araya (2011) is introduced to prescribe turbulent inflow conditions. Simulations are carried out for two different blowing ratios and film cooling effectiveness is calculated for both cases. Results obtained from LES will be compared with experimental results.

12:27PM H23.00010 Turbulence Structure and its Signature in Hypersonic Turbulent Boundary Layers , YIN CHIU KAN, PINO MARTIN, University of Maryland — 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 geometric packet identification techniques and statistical tools to track and study hairpin packets, as well as their wall signatures and their association with superstructures. In addition, we will use a spatio-temporal pattern finding process to track multiple packets evolutions concurrently.

Monday, November 19, 2012 10:30AM - 12:27PM –

Session H24 Aerodynamics III 30E - Chair: Feng Liu, University of California, Irvine

10:30AM H24.00001 Multiple Solutions of Transonic Flow over NACA0012 Airfoil , JUNTAO XIONG[1], YA LIU, FENG LIU[2], SHIJUN LUO, University of California, Irvine, ZIJIE ZHAO, XUDONG REN, CHAO GAO[3], Northwestern Polytechnical University — Multiple solutions of the small-disturbance potential equation and full potential equation were known for the NACA0012 airfoil in a certain range of transonic Mach numbers and at zero angle of attack. However the multiple solutions for this airfoil were not observed using Euler or Navier-Stokes equations under the above flow conditions. In the present work, both the Unsteady Reynolds-Averaged Navier-Stokes (URANS) computations and transonic wind tunnel experiments are performed under certain Reynolds numbers to further study the problem. The results of the two methods reveal that buffet appears in a narrow Mach number range where the potential flow methods predict multiple solutions. Boundary layer displacement thickness computed from URANS at the same flow condition is used to modify the geometry of the airfoil. Euler equations are then solved for the modified geometry. The results show that the addition of the boundary layer displacement thickness creates multiple solutions for the NACA0012 airfoil. Global linear stability analysis is also performed on the original and the modified airfoils. This shows a close relationship between the viscous unsteady shock buffet phenomenon of transonic airfoil flow and the existence of multiple solutions of the external inviscid flow.

[1]Postdoctoral Research Assistant
[2]Professor
[3]Professor

10:43AM H24.00002 Stability of Inviscid Flow over Airfoils Admitting Multiple Numerical Solutions , YA LIU[1], JUNTAO XIONG[2], FENG LIU[3], SHIJUN LUO[4], University of California, Irvine — Multiple numerical solutions at the same flight condition are found of inviscid transonic flow over certain airfoils (Jameson et al., AIAA 2011-3509) within some Mach number range. Both symmetric and asymmetric solutions exist for a symmetric airfoil at zero angle of attack. Global linear stability analysis of the multiple solutions is conducted. Linear perturbation equations of the Euler equations around a steady-state solution are formed and discretized numerically. An eigenvalue problem is then constructed using the modal analysis approach. Only a small portion of the eigen spectrum is needed and thus can be found efficiently by using Arnoldi's algorithm. The least stable or unstable mode corresponds to the eigenvalue with the largest real part. Analysis of the NACA 0012 airfoil indicates stability of symmetric solutions of the Euler equations at conditions where buffet is found from unsteady Navier-Stokes equations. Euler solutions of the same airfoil but modified to include the displacement thickness of the boundary layer computed from the Navier-Stokes equations, however, exhibit instability based on the present linear stability analysis.

[1]Graduate Student
[2]Postdoctoral Researcher
[3]Professor
[4]Researcher

10:56AM H24.00003 Parametric study on separation control by DBD plasma actuator over NACA0012 and NACA0015 airfoil at Reynolds number 63,000 , MAKOTO SATO, TAKU NONOMURA, HIKARU AONO, Institute of Space and Astronautical Science, JAXA, KOICHI OKADA, Ryoyu Systems Co., Ltd, KOZO FUJII, Institute of Space and Astronautical Science, JAXA — Large-eddy simulations of the separated flow over NACA0012 and NACA0015 airfoil, which are controlled by a DBD plasma actuator, are conducted to clarify the relationship between turbulent transition around the airfoil and aerodynamic performance. In these simulations, position and operation conditions of DBD plasma actuator are varied as simulation parameters. The install position of actuator is 0 % , 2.5 %, 5 % and 10 % chord length from the leading edge. The burst frequency is changed from 0.5 to 20. In addition, the degree of induced flow and burst ratio of actuation are changed. The promotion of turbulent transition around airfoil is closely related to the control of separation. From the parametric study of DBD position, it is clarified that the effective position of actuator to suppress the separation is near the separation point. In especial, the upstream of separation position is better for further suppression. From the simple analyses of turbulent kinetic energy distributions, it is clearly observed that the cases with earlier and smooth turbulent transition over airfoil have better aerodynamic performance in almost cases.

11:09AM H24.00004 Separation Control by External Acoustic Excitation on a Finite Wing at Low Reynolds Numbers[1] , SHANLING YANG, GEOFFREY SPEDDING, University of Southern California — For Reynolds numbers between 10000 and 100000, many smooth airfoils have complex lift-drag polars that can include multiple states at single points in the control parameter, the angle of attack. The E387 experiences pre-stall hysteresis and abrupt switching between stable states that result from sudden flow reattachment and the formation of a laminar separation bubble. External acoustic excitation is shown to strongly modify the flow dynamics. Separation control, hysteresis elimination, and more than 70% increase in aerodynamic efficiency are obtained at select excitation frequencies and sound pressure levels. Flow reattachment and the appearance of vortical structures in the separated shear layer are achieved by acoustic forcing. Correlation between the effects from acoustic forcing and wind tunnel resonances shows that the anti-resonances in a closed chamber correspond to the largest improvement in wing performance. Further applications for the control and stabilization of small-scale aircraft both in and out of closed chambers are considered.

[1]This work was supported by AFOSR grant no. FA 9550-11-1-0106.

11:22AM H24.00005 ABSTRACT WITHDRAWN –

11:35AM H24.00006 On The Symmetry of Proper Orthogonal Decomposition Modes of a Flapping Foil , ZONGXIAN LIANG, HAIBO DONG, Wright State University — In this work, the proper orthogonal decomposition (POD) is applied to investigate the flow field generated by a finite-aspect-ratio flapping foil undergoing a pitching and plunging motion. It is found that geometrical symmetry of the flow field caused by geometrical symmetry of the foil can be preserved by spatial symmetry of POD modes, meanwhile a half-period symmetry caused by symmetric motion of the foil classifies the POD modes into two sets of symmetry patterns with respect to their frequencies. Relations between the symmetry patterns and the direction of aerodynamic forces are discussed.

11:48AM H24.00007 Vorticity forces on a delta wing from the perspective of a force element theory , CHENG-TA HSIEH, JIAN-JHIH LEE, CHIEN C. CHANG, CHIN-CHOU CHU, National Taiwan University — In this study, we consider various force contributions to an impulsively started delta wing from the perspective of a force element theory. A wing plate of aspect ratio AR is placed at the angle of attack (α) to the incident stream. We consider 3 aspect ratios of the delta wing: AR=1, 2, 4 and 3 angles of attack α=15°, 30° and 45°, while the Reynolds number Re is set to be the fixed 300. The force element theory enables us to examine forces exerted on the a delta wing credited to the individual flow structures, such as the leading-edge vortex (LEV), trailing edge vortex (TEV), as well as the contribution from the surface vorticity. It is widely known that flow over a delta wing is genuinely three-dimensional, and there is no two-dimensional analog. Here, we provide an insightful understanding of flow characteristics by relating the forces directly to the various sources of vorticity (wx, wy and wz) on or near the wing plate. The relative importance of the various vorticity contributions to the hydrodynamic forces is analyzed in terms of the aspect ratio and angle of attack.

12:01PM H24.00008 Hydrodynamic characteristics for flow around wavy wings[1] , MI JEONG KIM, Department of Naval Architecture and Ocean Engineering, Pusan National University, HYUN SIK YOON, Global core research center for ships and offshore plants, Pusan National University — The present study numerically investigates the effect of the wavy leading edge on the flow of rectangular wings with the low aspect ratio of 1.5 in wide range of the angle of attack ($0° \leq \alpha \leq 40°$) at one Reynolds number of 10^6. Five different wave amplitudes of 0.01C, 0.02C, 0.03C, 0.04C and 0.05C at fixed wavy length of S/8 have been considered, where C and S are the mean chord length of the wing and the span length, respectively. The stall angle is dependent on the wave amplitude. The smallest wave amplitude of 0.1C among the wave amplitudes considered in this study and the smooth wing revealed the similar variation of lift coefficient C_L according to α, resulting in the same stall angle of 20° where the maximum lift appears. As the wave amplitude increases from 0.01C to 0.02C, the stall angle became smaller as α=18°. When the wave amplitudes are 0.03C and 0.04C, the stall angle keeps maintaining with the same value of a =0.02C. When the wave amplitude is 0.05C which is the largest one among the wave amplitudes considered in this study, the earliest onset of the stall has been presented at α=16°. In the post-stall region, C_L for all of the wavy cases recovered and became almost the same as the smooth wing.

[1]This work was supported by the National Research Foundation of Korea (NRF) grant funded by the Korea government (MEST) through GCRC-SOP (No. 2011-0030662)

12:14PM H24.00009 Effects of Dynamic Pitching on Wind Turbine Blade Performance[1] , JONATHAN NAUGHTON, ASHLI BABBITT, JOHN STRIKE, MICHAEL HIND, ANDREW MAGSTADT, POURYA NIKOUEEYAN, University of Wyoming — Due to turbulence in the wind and the rotation of the blade through a shear layer, wind turbine blade flows are inherently unsteady. Over the past five years, a number of wind turbine blade sections used for inboard, mid-span, and tip regions of the blade (including flatback airfoils) have been tested at a Reynolds number of 225,000. The airfoils have been tested at reduced frequencies $c\omega/2U$, where c is the chord length, ω is the oscillation frequency (radians/sec), and U is the air velocity ahead of the blade, relevant to commercial wind turbines. Unsteady pressure measurements and Particle Image Velocimetry (PIV) have provided information about the surface properties and surrounding flow field and their relationship. The results have shown that, depending on the reduced frequency, a lag in pressure and flow-field structures is experienced by the blade. When the blade is operating at angles above the static stall angle, delayed separation is experience as expected. The reattachment of the flow is also delayed, and, at higher reduced frequencies, the flow can remain separated throughout the entire downward pitching movement. Such dynamic data result in a better understanding of the unsteady flow physics necessary for improved designs.

[1]Support from DOE and BP is acknowledged

Monday, November 19, 2012 10:30AM - 12:14PM –

Session H25 Flow Control: Internal flow 31A - Chair: Michael Amitay, Rensselaer Polytechnic Institute

10:30AM H25.00001 Feedback stabilization of vortex flows in a finite-length straight pipe , S. WANG, R. GONG, U Auckland, Z. RUSAK, L. XU, RPI, S. TAYLOR, L. JENG, U Auckland — The properties of a recently proposed[1] feedback stabilization method of swirling flows in a finite-length pipe are studied. In the natural case, when swirl is above a critical level, linearly unstable modes appear in sequence as swirl increases and evolve to a vortex breakdown state. Based on a long-wave approach, the feedback control methodology is shown to enforce decay of perturbation's kinetic energy and to quench all instability modes at above critical swirl. In the case of a solid-body rotation, the effectiveness of this control approach is further analyzed through a mode analysis of the full linearized flow control problem. We first establish the asymptotic decay of all modes with real growth rates. We then compute growth rates of all modes according to the linearized flow control problem for swirl up to 50% above critical level. Flow stabilization in the whole swirl range is demonstrated. However, control effectiveness is sensitive to choice of the control gain. An inadequate gain, either insufficient or excessive, could lead to a failure of control at high swirl levels. Predictions of controlled flow cases agree with numerical simulations using the full unsteady and axisymmetric Euler equations with fluidic actuation along the pipe wall.

[1]Rusak et al JFM 2012.

10:43AM H25.00002 Design of Servo-Driven Actuators for Spanwise-Varying Control of a Backward-Facing Step Flow[1] , MARC SCHOSTEK, LORENZ SIGURDSON, Vortex Fluid Dynamics Lab, University of Alberta — For an experimental study of a forced backward-facing step water flow the design of 16 piston actuators was necessary. The 16 actuators connect to manifolds to force the flow at the step edge through many more actuation ports. The 16 actuators allowed for variant forcing in the spanwise direction with a resolution of 0.5 times the step height h. They are capable of producing unique perturbation waveforms of forcing velocity amplitudes $0 < u'/U_\infty \leq 2$ and either single or multiple forcing Strouhal numbers in the range $0 < St_h \leq 1.0$. These forcing amplitudes are larger than ever used in any previous forced backward-facing step flow experiments. The process of designing the servo-motor driven actuator system will be discussed.

[1]Support from NSERC Grant 41747 is gratefully acknowledged.

10:56AM H25.00003 High Amplitude Forcing Dependence of Control of a Backward-Facing Step Flow[1] , LORENZ SIGURDSON, MARC SCHOSTEK, University of Alberta — An experimental study was conducted of a forced backward-facing step water flow, which required the design of 16 piston actuators. They allowed spanwise-varying forcing with a resolution of 0.5 times the step height h. They were capable of producing unique perturbation waveforms of forcing velocity amplitudes $0 < u'/U_\infty \leq 2$, and forcing Strouhal numbers based on h in the range $0 < St_h \leq 1.0$. These forcing amplitudes are larger than those used in known previous forced backward-facing step flow experiments. For measurement of the reattachment length a hydro-tuft was designed which indicated flow direction. A set of images taken of an array of tufts was processed to calculate a time-averaged reattachment line. Initial experiments were for spanwise-invariant forcing for the full amplitude range and forcing frequencies of $0 < St_h \leq 0.5$. The results showed an optimal St_h which shifted to a lower value with increasing forcing amplitude, and a non-monotonic shortening of the reattachment length. As a function of forcing amplitude, reattachment reached a pronounced local minimum at $u'/U_\infty \approx 0.3 - 0.4$, and then rose to a local maxima at a $u'/U_\infty \approx 0.5 - 0.6$. Larger forcing amplitudes caused even more shortening than the local minimum.

[1]Support from NSERC Discovery Grant 41747 is gratefully acknowledged.

11:09AM H25.00004 Feed-forward control of the flow over a backward-facing step , FABIEN JUILLET, Ecole Polytechnique, PETER SCHMID, CNRS - Ecole Polytechnique, BEVERLEY MCKEON, Caltech - GALCIT — In this study, the control of incoming perturbations in convection-dominated flows is analyzed numerically and experimentally. For this purpose, multiple sensors and actuators are used. First, a model is built from input and output data sequences using a least-squares system identification. Then, a feed-forward Model Predicitive Controller (MPC) is designed. It appears that feed-forward control is particularly relevant when applied to convection-dominated flows. A very general and flexible formulation of the technique is introduced and validated on the flow over a backward-facing step. Although the objective sensors are localized on the walls, the impact of the control is more global and perturbations are also reduced in the middle of the channel. The coupling of system identification together with feed-forward control was found to be a flexible, efficient and experimentally feasible strategy. In particular, the successful numerical control is further supported by experimental results. Support from Ecole Polytechnique and the Partner University Fund (PUF) is gratefully acknowledged.

11:22AM H25.00005 Experimental Investigation of Flow Control in a Compact Inlet Duct[1] , BRIAN DEBRONSKY, MICHAEL AMITAY, Rensselaer Polytechnic Institute — Attractive to aircraft designers are compact inlets, which implement curved flow paths from the air intake of the engine to the compressor face. A compromise must be made between the compactness of the inlet and its aerodynamic performance. The aerodynamic purpose of inlets is to decelerate the oncoming flow before reaching the engine while minimizing total pressure loss, unsteadiness and distortion. Low length-to-diameter ratio inlets have a high degree of curvature, which inevitably causes flow separation and secondary flows. To address this issue, active flow control was implemented on a compact (L/D = 1.6) inlet to improve its performance metrics. The experiments were conducted at a Mach number of 0.44, where the actuation from an array of skewed and pitched jets produced streamwise vortices opposite to the secondary flow structures. The actuation resulted in an improved pressure recovery at the aerodynamic interface plane (AIP), where both the strength of the secondary structures and the flow unsteadiness were significantly reduced.

[1]Northrop Grumman Corporation

11:35AM H25.00006 The valveless impedance pump and the unexpected effect of convection in tight spaces[1] , JAMES WOODCOCK, University of Melbourne, JOHN SADER, Department of Mathematics and Statistics, University of Melbourne, IVAN MARUSIC, Department of Mechanical Engineering, University of Melbourne — The valveless impedance pump (VIP) consists of a thin tube, one section of which is elastic and is subjected to rhythmic pinching at some point offset from its centre. This induces travelling waves which propagate back and forth along an elastic section of a tube, which in turn induces a flow within the tube. We have investigated the physics underlying the VIP using a perturbation analysis and found that, contrary to expectations, convection plays an important role despite the thinness of the tube. Using numerical simulations, it has been shown that convection can in fact be the dominant mechanism at work within the flow if the oscillations of the tube are of sufficient amplitude. We propose that convection may generally play an important role, even within thin tubes and channels, where the velocity gradients along the wall are significant.

[1]The authors wish to acknowledge the financial support of the Australian Research Council and the University of Melbourne Postgraduate Scholarship Scheme.

11:48AM H25.00007 Transient Dynamics Modeling of Experimental Hypersonic Inlet Unstart[1] , KELLEY E. HUTCHINS, MICHAEL SZMUK, NOEL T. CLEMENS, MARUTHI R. AKELLA, The University of Texas at Austin, JEFFREY M. DONBAR, Air Force Research Laboratory, Wright-Patterson AFB, SIVARAM GOGINENI, Spectral Energies, LLC — During unstart, the rapid upstream propagation of a hypersonic engine's inlet-isolator shock system can be readily detected through pressure measurements. Specifically, the magnitude of the pressure readings suddenly and dramatically increases as soon as the leading edge of the shock system passes the measurement location. In this work, attempts to model the transient dynamics governing shock motion have been made through the use of system identification techniques. The result of these efforts is a partially nonlinear dynamic model that describes shock motion through pressure signals. The process reveals the possibility of partitioning the nonlinear behaviors from the linear dynamics with relative ease. Related attempts are then made to create a model where the nonlinear portion has been pre-specified leaving only the linear portion to be determined by system identification. The modeling and identification process specific to the unstart data used is discussed, and successful models are presented for both the full system identification and the partitioned model cases. The suitability of various input data types is explored, and comments on practicality are made.

[1]This work is supported in part by AFRL under SBIR contract

12:01PM H25.00008 Closed-loop control of a turbulent mixing layer - experimental study[1] , VLADIMIR PAREZANOVIC, JOEL DELVILLE, CARINE FOURMENT, LAURENT CORDIER, BERND NOACK, Institute PPRIME, France, TAMIR SHAQARIN, Tafila Technical University, Jordan — Open and closed-loop control of a turbulent mixing layer is experimentally performed in a dedicated wind-tunnel facility (TUCOROM). The flow is manipulated with 100 independently operating fluidic micro-valve actuators integrated transversely in the trailing edge of the splitter plate. Sensing is performed with a rake of 30 hot-wire probes located downstream in the mixing layer. The control goal is a manipulation of the spreading rate. The underlying physical mechanisms employ a wide range of frequencies as well as a wide range of spanwise modes. The calculated Reynolds number based on vorticity thickness is about $Re = 2000$. Control authority is presented with PIV and hot-wire results.

[1]Partially funded by the ANR Chair of Excellence TUCOROM.

Monday, November 19, 2012 10:30AM - 12:40PM –

Session H26 Reactive Flows V: Numerical Approaches to Turbulent Combustion 31B - Chair: Venkat Raman, University of Texas at Austin

10:30AM H26.00001 A novel methodology for simulating low Mach number combustion , AMMAR ABDILGHANIE, JAMES RILEY, University of Washington, OSCAR FLORES, Universidad Carlos III de Madrid, ROBERT MOSER, University of Texas at Austin — The velocity-vorticity formulation for the solution of unsteady three-dimensional incompressible flows (Kim, Moin and Moser JFM 1987) is extended for compressible flows under the low Mach number approximation for combustion applications. A key advantage of the methodology is the elimination of the pressure from the momentum equations and, as a result, the errors and complications of pressure boundary conditions associated with pressure-splitting algorithms. The added efficiency of the method for horizontally homogeneous flows makes it computationally very attractive. The hydrodynamic part of the algorithm comprises two evolution equations for two dependent variables and two Poisson-type equations that, by construction, ensure mass conservation. Sixth-order accurate compact scheme is used for spatial discretization in the vertical and third order implicit-explicit Runge-Kutta is used for time advancement. Open boundary conditions are used at the top boundary and free-slip no-flux conditions are employed at the bottom. Simulations of compressible Taylor Green vortex flow are briefly discussed and future research directions are summarized.

10:43AM H26.00002 Vorticity dynamics in variable density flows , PETER HAMLINGTON, University of Colorado, Boulder, ALEXEI POLUDNENKO, ELAINE ORAN, Naval Research Laboratory — The dynamics of vorticity in incompressible flows has been the subject of considerable research, but remains relatively poorly understood in variable density flows. Such flows include reacting and supersonic flows where the behavior of the vorticity is central to understanding the interactions between turbulence, shock waves, and flames. The variations in density across shocks and flames are often anisotropic, and here we discuss the differing effects of isotropic and anisotropic density changes on the vorticity. We focus in particular on flames and shocks, which represent different ends of the variable density spectrum; flames can create a rapid anisotropic expansion of the fluid while shocks produce a rapid anisotropic compression. These density changes result in anisotropic vorticity suppression across flames and anisotropic vorticity generation across shocks. In reacting flows, we discuss the effects of anisotropic suppression on intermittency, turbulence-flame interactions, and flame properties. We also propose a decomposition of the strain rate that allows the relative effects of turbulent and flame straining to be understood. We then compare vorticity-shock interactions with the reacting flow case and outline directions for future research.

10:56AM H26.00003 Discontinuous Galerkin Method for Combustion , YU LV, MATTHIAS IHME, Department of Aerospace Engineering, University of Michigan — Over recent years, the discontinuous Galerkin (DG) method has found increased interest in application to advection-dominated hyperbolic flows. However, extending DG to combustion introduces several challenges that are associated with the treatment of diffusion-dominated multi-species transport, non-uniform thermal properties, and the consideration of complex and stiff reaction chemistry. By considering all of these aspects, a DG-formulation for multi-species combustion has been developed. After presenting the numerical method and demonstrating higher-order convergence properties, this formulation is applied to relevant combustion problems, involving multi-species mixing, deflagration, and detonation-systems. Results are compared against solutions from conventional finite-volume/finite-difference-schemes, and potential benefits of the DG-method for combustion are discussed.

11:09AM H26.00004 Empirical low-dimensional manifolds in composition space[1] , YUE YANG, STEPHEN B. POPE, Cornell University, JACQUELINE H. CHEN, Sandia National Laboratories — To reduce the computational cost of turbulent combustion simulations with a detailed chemical mechanism, it is useful to find a low-dimensional manifold in composition space that can approximate the full system dynamics. Most previous low-dimensional manifolds in turbulent combustion are based on the governing conservation equations or thermochemistry and their application involves certain assumptions. On the other hand, empirical low-dimensional manifolds (ELDMs) are constructed based on samples of the compositions observed in experiments or in direct numerical simulation (DNS). Plane and curved ELDMs can be obtained using principal component analysis (PCA) and multivariate adaptive spline regression (MARS), respectively. Both PCA and MARS are applied to the DNS datasets of a non-premixed CO/H2 temporally evolving jet flame (Hawkes et al., 2007) and an ethylene lifted jet flame (Yoo et al., 2011). We observe that it requires very high dimensions to represent the species mass fractions accurately by a plane ELDM, while better accuracy can be achieved by curved ELDMs with lower dimensions. In addition, the effect of differential diffusion on ELDMs is examined in large-eddy simulations with PDF modeling.

[1]This work is supported in part by the Combustion Energy Frontier Research Center funded by the DOE

11:22AM H26.00005 An Adjoint Approach for Determining Sensitivity of Combustion Simulations to Model Parameters , KALEN BRAMAN, VENKAT RAMAN, The University of Texas at Austin Dept of Aerospace Engineering and Engineering Mechanics — Simulations of turbulent combustion typically involve numerous models including those for the gas phase chemistry, turbulence, and combustion. Such models typically involve a host of parameters, and simulation results are generally sensitive to these parameters. Here, an adjoint-based approach is developed to determine the sensitivity of emissions at the combustion exit to these model parameters. First, adjoint equations for the turbulent combustion system in the context of the Reynolds-averaged Navier-Stokes (RANS) approach are derived. Then, a novel numerical scheme for solving these equations is introduced. The methodology is verified by comparing against a forward sensitivity computation. Finally, the sensitivity of emissions to model parameters is determined in a canonical jet flame configuration.

11:35AM H26.00006 Relating filtered and unfiltered quantities in large eddy simulation of turbulent combustion , VENKATRAMANAN RAMAN, COLIN HEYE, The University of Texas at Austin — Large eddy simulation (LES) is currently recognized as a valuable tool for modeling turbulent combustion. Although significant advances have been made in the development of reliable models in LES, there remains considerable ambiguity regarding the fundamental definition of the methodology and its use for predicting measurable flow quantities. The root cause of these problems is the filtering operation used to obtain the LES governing equations. Here, we discuss the relation between filtered and unfiltered quantities through a probabilistic framework. Basic transformation rules to relate combustion-related quantities to mixture fraction or other such scalars are presented. Using a direct numerical simulation (DNS) of homogeneous isotropic turbulence, the importance of a missing modeling step is discussed.

11:48AM H26.00007 PDF investigations of turbulent non-premixed jet flames with thin reaction zones , HAIFENG WANG, Purdue University, STEPHEN POPE, Cornell University — PDF (probability density function) modeling studies are carried out for the Sydney piloted jet flames. These Sydney flames feature much thinner reaction zones in the mixture fraction space compared to those in the well-studied Sandia piloted jet flames. The performance of the different turbulent combustion models in the Sydney flames with thin reaction zones has not been examined extensively before, and this work aims at evaluating the capability of the PDF method to represent the thin turbulent flame structures in the Sydney piloted flames. Parametric and sensitivity PDF studies are performed with respect to the different models and model parameters. A global error parameter is defined to quantify the departure of the simulation results from the experimental data, and is used to assess the performance of the different set of models and model parameters.

12:01PM H26.00008 Modeling local extinction in turbulent combustion using an embedding method , ROBERT KNAUS, CARLOS PANTANO, University of Illinois at Urbana-Champaign — Local regions of extinction in diffusion flames, called "flame holes," can reduce the efficiency of combustion and increase the production of certain pollutants. At sufficiently high speeds, a flame may also be lifted from the rim of the burner to a downstream location that may be stable. These two phenomena share a common underlying mechanism of propagation related to edge-flame dynamics where chemistry and fluid mechanics are equally important. We present a formulation that describes the formation, propagation, and growth of flames holes on the stoichiometric surface using edge flame dynamics. The boundary separating the flame from the quenched region is modeled using a progress variable defined on the moving stoichiometric surface that is embedded in the three-dimensional space using an extension algorithm. This Cartesian problem is solved using a high-order finite-volume WENO method extended to this nonconservative problem. This algorithm can track the dynamics of flame holes in a turbulent reacting-shear layer and model flame liftoff without requiring full chemistry calculations.

12:14PM H26.00009 Numerical simulation of turbulent stratified flame propagation in a closed vessel , CATHERINE GRUSELLE, CORIA CNRS UMR 6614 / Renault, GHISLAIN LARTIGUE, CORIA CNRS UMR 6614, PERRINE PEPIOT, Cornell University, VINCENT MOUREAU, YVES D'ANGELO, CORIA CNRS UMR 6614 — Reducing pollutants emissions while keeping a high combustion efficiency and a low fuel consumption is an important challenge for both gas turbine (GT) and internal combustion engines (ICE). To fulfill these new constraints, stratified combustion may constitute an efficient strategy. A tabulated chemistry approach based on FPI combined to a low-Mach number method is applied in the analysis of a turbulent propane-air flame with equivalence ratio (ER) stratification, which has been studied experimentally by Balusamy [S. Balusamy, Ph.D Thesis, INSA-Rouen (2010)]. Flame topology, along with flame velocity statistics, are well reproduced in the simulation, even if time-history effects are not accounted for in the tabulated approach. However, these effects may become significant when exhaust gas recirculation (EGR) is introduced. To better quantify them, both ER and EGR-stratified two-dimensional flames are simulated using finite-rate chemistry and a semi-detailed mechanism for propane oxidation. The numerical implementation is first investigated in terms of efficiency and accuracy, with a focus on splitting errors. The resulting flames are then analyzed to investigate potential extensions of the FPI technique to EGR stratification.

12:27PM H26.00010 Evaluation of a Consistent LES/PDF Method Using a Series of Experimental Spray Flames , COLIN HEYE, VENKAT RAMAN, The University of Texas at Austin — A consistent method for the evolution of the joint-scalar probability density function (PDF) transport equation is proposed for application to large eddy simulation (LES) of turbulent reacting flows containing evaporating spray droplets. PDF transport equations provide the benefit of including the chemical source term in closed form, however, additional terms describing LES subfilter mixing must be modeled. The recent availability of detailed experimental measurements provide model validation data for a wide range of evaporation rates and combustion regimes, as is well-known to occur in spray flames. In this work, the experimental data will used to investigate the impact of droplet mass loading and evaporation rates on the subfilter scalar PDF shape in comparison with conventional flamelet models. In addition, existing model term closures in the PDF transport equations are evaluated with a focus on their validity in the presence of regime changes.

Monday, November 19, 2012 10:30AM - 12:14PM –

Session H27 Separated Flows III 31C - Chair: Sanjay Kumar, University of Texas at Brownsville

10:30AM H27.00001 A numerical study of the unfluence of wall effects on the onset of unsteadiness in the three dimensional flow over a backward-facing step , NIKOLAOS MALAMATARIS, George Mason University - TEI of W.Macedonia — In three dimensional separated flows, a flow component is developed in the spanwise direction that permits the flow to laterally escape [1]. This work shows for the first time how this flow situation occurs in the three dimensional, backward-facing step flow in a numerical experiment that mimics actual laboratory conditions (expansion ratio 1:2, aspect ratio 1:40). To this purpose, the full three dimensional Navier Stokes equations are solved directly with finite elements up to the highest Reynolds number (Re = 950) where the flow is stable. The wall effects are studied thoroughly, by showing how the recirculation regions vary close to the lateral wall and how the limiting streamlines are related to the spanwise flow in terms of their direction depending on the magnitude of the Reynolds number. It is shown how this spanwise flow goes all the way to the lateral wall and bounces back in a manner that it is impossible to be sustained at Reynolds numbers higher than 950 [2]. It is argued that this flow is responsible for the early onset of unsteadiness for this flow as has been observed in laboratory experiments and never fully understood so far.

[1] J.Délery, Onera on-line lessons, Part I, 2011
[2] N.A.Malamataris, DOI: 10.1002/fld.3699

10:43AM H27.00002 Direct Numerical Simulation of laminar separation bubbles , O.N. RAMESH, SAURABH PATWARDHAN, ABHIJIT MITRA, Indian Instiute of Science — This work presents the DNS of laminar separation bubbles (LSB) that formed over a flat plate due to an imposed pressure gradient. Mean flow parameters such as mean velocity, static pressure distribution and the geometric parameters, such as aspect ratio of the LSB, over the plate closely corresponds to those found in experiments and literature. The locus of the inflection point of the mean velocity profile was found to lie outside the dividing streamline and this is expected to correspond to a convectively unstable bubble. A closer look of the LSB as when advects along the reverse flow streamline adjacent to the wall suggest that turbulence progressively decayed as one moved upstream. This is indicative of the phenomenon similar to relaminarisation in this region, presumably due to the decrease in pressure along the reverse flow streamline. The energy budget inside the dividing streamline showed interesting trends and these will be discussed during the presentation. Furthermore, the dynamics of free shear layer and nonlinearity will also be presented.

10:56AM H27.00003 Negative production of turbulent kinetic energy in a turbulent separation bubble , HIROYUKI ABE, YASUHIRO MIZOBUCHI, YUICHI MATSUO, Japan Aerospace Exploration Agency, PHILIPPE R. SPALART, Boeing Commercial Airplanes — DNS data are used to examine the behavior of turbulence in the boundary layer separating from a flat plate, and reattaching. Particular attention is given to a region of negative production of turbulent kinetic energy. The inlet Reynolds number R_θ based on momentum thickness is equal to 300, 600 and 900. In all cases, the production P_k is weak across the bubble and goes negative with a smaller magnitude than the dissipation at the top, where the streamline curvature is convex. An indicator of streamwise curvature $U_{2,1}$, which comes from a rapid pressure-driven change of the mean strain rate, is indeed associated with negative P_k. That is, the budget term arising from $U_{2,1}$ yields negative Reynolds shear stress ($-\overline{uv} < 0$), and then the product of $-\overline{uv}$ and $U_{1,2}$ contributes to negative P_k. There is no one-to-one correspondence in a region between negative $-\overline{uv}$ and negative P_k. The correspondence is however excellent when the Reynolds shear stress is defined in the streamline orthogonal coordinate system, i.e., $\overline{ab} \equiv \left((\overline{vv} - \overline{uu})U_1U_2 + \overline{uv}(U_1^2 - U_2^2)\right) \Big/ (U_1^2 + U_2^2)$, which underlines that the streamline curvature is an important ingredient for negative P_k.

11:09AM H27.00004 Direct measurement of wall shear stress in a backward facing step flow by using a photonic wall shear stress sensor , ULAS AYAZ, TINDARO IOPPOLO, VOLKAN OTUGEN, Southern Methodist University — We report direct wall shear stress measurements in a reattaching channel flow. The sensor used to perform measurements is a photonic wall shear stress sensor based on the morphology dependent resonances (MDR) of dielectric microspheres. The wall shear stress acting on a circular movable plate with 1 mm diameter, is mechanically transmitted to a Polydimethylsyloxane (PDMS) microsphere. The applied shear force on the microsphere leads to a shift in the MDRs, thus, by monitoring the MDR shifts, the magnitude as well as the direction of the shear stress are measured. The sensor is calibrated in a two dimensional channel with air flow. For flow separation and reattachment, a backward facing step is introduced into the channel and shear stress measurements have been performed at various distances from the step. Frequency and the magnitude of the shear stress fluctuations at the reattachment region have been recorded and compared to the reported measurements in literature.

11:22AM H27.00005 Unsteady separation in a forward-facing step flow[1] , DAVID PEARSON, Imperial College London, PAUL GOULART, ETH Zurich, BHARATHRAM GANAPATHISUBRAMANI, University of Southampton, IMPERIAL COLLEGE LONDON TEAM, ETH ZURICH TEAM, UNIVERSITY OF SOUTHAMPTON TEAM — The structure and behaviour of the separation region upstream of a forward step is investigated using time-resolved 2D Particle Image Velocimetry. Conditional averages of the flow-field based on the amount of reverse flow present are used to determine the shape and size of the separated flow in relation to the separation point. It is shown that the separation is of 'open' form with no reattachment point for approximately 50% of the time. When a reattachment point forms on the step face the separation region can become unstable and expand up and over the step corner. This transfer of mass occurs approximately 10% of the time and is postulated to be caused by large-scale transverse motions at the step face. The conditional averages can be traced backward in time to investigate the upstream flow field prior to such events. It is found that the large scale separations are preceded by a region of low momentum flow convecting toward the step. This momentum deficit creates the conditions under which the separation expands. The size and shape of the momentum deficit, and the timescales over which it acts, is consistent with the large boundary layer structures observed in the literature.

[1]EPSRC, through grant no. EP/F056206/1 and EU-FP7 through Grant No. FP7-ICT-2009-4248940

11:35AM H27.00006 Evolution of coherent vortical structures in turbulent flow over backward-facing step , PANKAJ NADGE, RAGHURAMAN GOVARDHAN, Indian Institute of Science, Bangalore — The flow over a backward-facing step represents a geometrically simple flow that exhibits both boundary layer separation and reattachment. In the present work, we use detailed PIV measurements downstream of the step to help understand the evolution of vortical structures in this flow. In particular, velocity field measurements are done in a plane parallel to the lower wall (streamwise-spanwise plane). Upstream of the step, instantaneous velocity fields in this plane show counter-rotating vortical structures that are signatures of the three dimensional hairpin vortex structures present in turbulent boundary layers. These counter-rotating structures can be identified by the low streamwise velocity that exists between them. Conditional averaging of velocity fields gives clear counter-rotating structures whose length-scales can be measured. For the present case of a backward-facing step, the flow is evolving with streamwise distance after the step. Using similar techniques, we identify such counter-rotating structures downstream of the step; starting from near the step in the separated shear layer, all the way until well after shear layer reattachment on to the lower wall. Details about these structures and their evolution with streamwise distance will be presented at the conference.

11:48AM H27.00007 Experimental investigation of the leading edge vortex on vertical axis wind turbine blades[1] , REEVE DUNNE, BEVERLEY MCKEON, California Institute of Technology — A NACA 0018 airfoil is pitched about the leading edge over a large angle of attack range ($\pm \sim 40^\circ$) at a chord Reynolds number of 110,000 to simulate the flow over a single blade in a vertical axis wind turbine (VAWT). Particle image velocimetry (PIV) measurements are made to investigate the effects of pitching on leading edge vortex (LEV) development and separation. Time resolved experiments are performed to track vortex formation and convection over the airfoil for sinusoidal pitching motions corresponding to a VAWT trajectory as well as impulsive pitch up and pitch down motions. These results are compared to the wake of steady, post stall, high angle of attack airfoils ($\alpha = 20^\circ - 30^\circ$). The characteristics of the leading edge vortex development and subsequent separation from the airfoil are discussed, with a view to characterizing its effect on power generation with VAWTs and future flow control strategies for turbine performance improvement

[1]Funding from the Gordon and Betty Moore Foundation is gratefully acknowledged.

12:01PM H27.00008 Unsteady Cavity Induced Vibrations of Flexible Hydrofoils , DENIZ TOLGA AKCABAY, EUN JUNG CHAE, YIN LU YOUNG, Department of Naval Architecture and Marine Engineering, University of Michigan — The objective of this study is to investigate the dynamic interplay between elastic foil deformation and unsteady sheet/cloud cavitation. Recently, there is increasing interest in the use of light and flexible materials in marine propulsion devices and controlled surfaces, which can deform/vibrate due to un-intentional overload when operating in off-design conditions, or due to intentional passive/active controlled response of the hydrofoil. Numerical studies are conducted by applying a new hybrid coupling approach to efficiently and stably couple a URANS solver with a simplified structural model of the cantilevered, rectangular foil represented by a two degree-of-freedom system. The numerical model is first validated with experimental measurements of a rigid and a plastic NACA 66 hydrofoil. Next, numerical results are shown for plastic NACA 66 hydrofoil with varying mass and stiffness in turbulent subcavitating and cavitating flows. The influence of varying mass and stiffness on the cavitation patterns, vorticity contours and flow streamlines, bending and twisting deformation, and hydrodynamic load coefficients are presented. In particular, results are shown for un-locked and locked-in response due to unsteady sheet/cloud cavitation induced vibration.

Monday, November 19, 2012 10:30AM - 12:40PM –

Session H28 Waves II 32A - Chair: James Duncan, University of Maryland

10:30AM H28.00001 Internal Wave Attractors: Topographic effects on wave reflection and energy propagation , ROB SUTTON, STUART DALZIEL, Cambridge University — It is well known that the angle of propagation of internal gravity waves depends on the ratio of the frequency of the waves to the buoyancy frequency of the stratification. This constraint on the direction of propagation causes the wavelength of an internal wave to either increase or decrease when it reflects from a sloping boundary. Such geometric focusing can lead to wave energy being enhanced in predictable regions of the ocean, giving rise to the possibility of an internal wave attractor where the energy is focused onto a limit cycle. In this paper we revisit internal wave attractors in a simple trapezoidal tank, exploring how the structure of the attractor is disrupted by replacing the sloping boundary with a staircase configuration comprising only horizontal and vertical surfaces. Simple ray tracing suggests an attractor cannot form in such a geometry despite the macroscopic shape remaining unchanged. We explore this configuration experimentally, varying the length scale of the individual steps, demonstrating an evolution towards the classical attractor as the step size is decreased.

10:43AM H28.00002 A Cloak of Invisibility Against Ocean Waves , REZA ALAM, University of California, Berkeley — In this talk we show that that floating objects in stratified fluids can be cloaked against broadband incident waves by properly architecting the bottom corrugations. The density of water in an ocean or a sea is typically not constant due to, mainly, variations of temperature and salinity. Stratified waters, besides regular surface waves, admit the so-called internal waves, which are gravity waves that propagate within the body of the water. The concept behind the presented scheme is based on nonlinear resonance of surface and interfacial waves with the bottom topography and is obtained due to the dispersive nature of gravity waves. Perfect cloaking against monochromatic waves can theoretically be achieved and was further investigated via a direct high-order spectral scheme. The presented cloak is the alignment of bottom corrugations only, and therefore is surface noninvasive. Cloaking in seas by bottom modifications may play a role in protecting near shore or offshore structures (buoys) and in creating shelter for fishermen during storms. In reverse it can result in disappearance and appearance of surface waves in areas where sandbars (or any other appreciable bottom variations) exist.

10:56AM H28.00003 ABSTRACT WITHDRAWN –

11:09AM H28.00004 An extended application for strongly nonlinear two-layer model[1] , SHENGQIAN CHEN, ROBERTO CAMASSA, University of North Carolina at Chapel Hill, Mathematics, WOOYOUNG CHOI, New Jersey Institute of Technology, Mathematics, ROXANA TIRON, University College Dublin, School of Mathematical Sciences — Strongly nonlinear internal wave models that have been developed in recent years have mostly been derived under long-wave, shallow-water assumptions. This talk will focus on assessing the applicability of these models in setups that go beyond their derivation hypotheses by comparisons with direct numerical simulations of near two-layer Euler-fluids' motion emanating from experimentally realizable initial conditions. By placing numerical filters that effectively truncate high Fourier modes, the ill-posedness associated with the model equations is resolved, allowing numerical time evolution studies to proceed. As wave profiles change dynamically, the numerical filters are designed adaptively. Compared with full Euler solutions, model evolutions show good agreement from small to moderate amplitude waves. Even for large amplitudes, when Kelvin-Helmholtz instability can occur, the primary waves are still captured in both amplitude and phase, demonstrating how an accurate filter implementation is capable of enhancing the models' predictive validity. Comparisons with two-layer Korteweg-de Vries (KdV) equation for the same initial conditions will also be presented, and advantages and shortcomings of the different models will be discussed.

[1]This research is supported by NSF RTG DMS-0502266, NSF RTG DMS-0943851, NSF RAPID CBET-1045653, NSF CMG ARC-1025523, NSF DMS-1009750, and ONR DURIP N00014-09-1-0840.

11:22AM H28.00005 Bottom Boundary Layer Turbulence under an Internal Solitary Wave of Depression: Effects of Barotropic Current , TAKAHIRO SAKAI, PETER DIAMESSIS, Cornell University, GUSTAAF JACOBS, San Diego State University — The development of the bottom boundary layer (BBL) in the footprint of an internal solitary wave of depression is investigated by means of spectrally accurate numerical simulations in two and three dimensions. The focus is on the subsequent generation and ejection of vortices that potentially leads to bed sediment resuspension. Our preliminary study, based on two-dimensional simulations, has indicated that introduction of a barotropic current enables the formation of a robust separation bubble in a region of adverse pressure gradient over the bed. As a result, the production of packets of instability waves from the separation bubble, which evolve into vortices intermittently shed into the water column, is greatly enhanced. In the present study, the effect of barotropic current is investigated in detail for different wave amplitudes and stratification profiles with the wave Reynolds number up to the order of 10^5. This parametric study is performed in idealized, two-dimensional flow fields, in which fully-nonlinear, fully-nonhydrostatic wave field obtained by solving the DJL equation is adopted as a base state flow. Some preliminary results will be presented from three-dimensional Large Eddy Simulations which have been performed for a select set of governing parameters to explore the transition into turbulence in the BBL.

11:35AM H28.00006 Waves and Currents: Hawking radiation in the hydraulics laboratory? , GREGORY LAWRENCE, Department of Civil Engineering, University of British Columbia, SILKE WEINFURTNER, Department of Physics and Astronomy, University of British Columbia, EDMUND TEDFORD, Department of Civil Engineering, University of British Columbia, MATTHEW PENRICE, WILLIAM UNRUH, Department of Physics and Astronomy, University of British Columbia — Laboratory experiments were performed to test an analogy between Hawking radiation (the process by which black holes radiate energy) and the propagation of water waves against an adverse current. A streamlined obstacle was placed in a flume to create a region of high velocity. Long waves generated downstream of the obstacle were blocked by this region and converted to a pair of short waves. The group velocities of both the converted waves were downstream, but one of the converted waves retained an upstream phase velocity, whereas the other had a downstream phase velocity. These waves are shown to be analogous to Hawking radiation.

11:48AM H28.00007 Spatio-temporal characterization of Capillary Wave Turbulence , MICHAEL BERHANU, ERIC FALCON, MSC, Paris Diderot University — Wave Turbulence concerns the study of the statistical properties of a set of numerous non-linear interacting waves. The archetype of this phenomenon are waves on the surface of a fluid [1]. We report a space-time characterization of Capillary Wave Turbulence, produced in the laboratory at the air-water interface. The three-dimensional shape of the free interface is measured as a function of time, by using an optical method, the "Diffusing Light Photography" [2], associated with a fast camera. Linear and non-linear dispersion relations are extracted by the computation of the spatio-temporal power spectrum of wave amplitude. When Wave Turbulence regime is reached, we observe power-law spectra both in time and in space, whose exponents are in agreement with the theoretical predictions of Wave Turbulence theory [3].

[1] A. C. Newell and B. Rumpf, Annu. Rev. Fluid Mech. 43, 59 (2011) [2] W. B. Wright, R. Budakian and S. J. Putterman, Phys. Rev. Lett. 76, 4528 (1996) [3] V. Zakharov, V. L'vov and G. Falkovich (1992) "Kolmogorov spectra of turbulence" Springer-Verlag

12:01PM H28.00008 Direct numerical simulations of gravity-capillary wave turbulence , LUC DEIKE, Univ Paris Diderot, Sorbonne Paris Cité, MSC, UMR 7057 CNRS, F-75 013 Paris, France, EU, DANIEL FUSTER, CNRS, UMR 7190, Institut Jean Le Rond d'Alembert, F-75005 Paris, France, EU, MICHAEL BERHANU, ERIC FALCON, Univ Paris Diderot, Sorbonne Paris Cité, MSC, UMR 7057 CNRS, F-75 013 Paris, France, EU — Direct numerical simulation of the full two phase Navier-Stokes equations, including surface tension are performed, using the code Gerris (Popinet, 2009), in order to investigate gravity-capillary wave turbulence. Wave turbulence concerns the study of the statistical and dynamical properties of a set of nonlinear interacting waves (Zakharov, 1992). Waves at the air-water interface, initially at rest, are excited at low wave-numbers and a stationary wave turbulence state is obtained after a time long enough (typically 30 periods of the wave forcing period). The space-time wave height power spectrum is calculated for both capillary and gravity waves regimes. The observed dispersion relation is in agreement with the theoretical one for linear gravity-capillary wave. The wave height power spectrum in the wave-number-space or in the frequency-space exhibit a power law and will be discussed with respects of weak turbulence theory (Zakharov, 2012). Finally the scaling of the spectrum with the injected power will be compared with theoretical and experimental works.

12:14PM H28.00009 An Experiment on Two-Dimensional Interaction of Solitary Waves in Shallow Water System , HIDEKAZU TSUJI, KEI YUFU, KENJI MARUBAYASHI, Research Institute for Applied Mechanics, Kyushu University Japan — The dynamics of solitary waves in horizontally two-dimensional region is not yet well understood. Recently two-dimensional soliton interaction of Kadmotsetv-Petviashvili (KP) equation which describes the weakly nonlinear long wave in shallow water system has been theoretically studied (e.g. Kodama (2010)). It is clarified that the "resonant" interaction which forms Y-shaped triad can be described by exact solution. Li et al. (2011) experimentally studied the reflection of solitary wave at the wall and verified the theory of KP equation. To investigate more general interaction process, an experiment in wave tank using two wave makers which are controlled independently is carried out. The wave tank is 4m in length and 3.6 m in width. The depth of the water is about 8cm. The wavemakers, which are piston-type and have board about 1.5m in length, can produce orderly solitary wave which amplitude is 1.0-3.5cm. We observe newly generated solitary wave due to interaction of original solitary waves which have different amplitude and/or propagation direction. The results are compared with the aforementioned theory of KP equation.

12:27PM H28.00010 ABSTRACT WITHDRAWN –

Monday, November 19, 2012 10:30AM - 12:14PM –

Session H29 Porous Media Flows V 32B - Chair: Paulo Arratia, University of Pennsylvania

10:30AM H29.00001 Impacts of Transport Properties of Porous Corrosion Product Layer on Effective Corrosion Rate , XIAOBAI LI, DAVID COOK, Bosch Research and Technology Center North America — Condensing exhaust gases containing H_2O, SO_3 and NO_x cause serious corrosion failure in various industry processes. For example, in modern compact heat cells, corrosion products deposit on top of the heat exchanger cooling fins, blocking the flow passages and drastically decreasing system performance. The transport properties of porous corrosion product layers play important role in determining the corrosion tendency and observed corrosion rate. To understand the corrosion mechanism for Aluminum alloy in sulfuric acid environment, impacts of transport properties of corrosion residual layers are investigated with different numerical models for porous layer diffusivity. The effective corrosion rates resulted from these models are compared to corresponding experimental measurements. A multilayer diffusivity model in which diffusivity depends both on porous layer structure and composition shows excellent agreements with experimental data. This model is currently being used in a multi-scale flow simulation framework to predict corrosion phenomena in heat cells.

10:43AM H29.00002 Overlimiting current and water purification in porous materials , DAOSHENG DENG, Department of Chemical Engineering, MIT, WASSIM AOUAD, Department of Chemical Engineering, École Polytechnique de Montréal, Canada, SVEN SCHLUMPBERGER, Department of Chemical Engineering, MIT, MARTIN Z. BAZANT, Department of Chemical Engineering and Mathematics, MIT — Salt transport in bulk electrolytes occurs by diffusion and convection, but in microfluidic devices and porous media, the presence of charged side walls leads to additional surface transport mechanisms, surface conduction and electro-osmotic flows, which become more important as the bulk salt concentration decreases. As a result, it is possible to exceed the diffusion-limited current to a membrane or electrode. In this work, we present experimental observations of over-limiting current to an ion-exchange membrane through a porous glass frit with submicron pores. Under this operation conditions, we also demonstrate the continuous extraction of depleted solution for water purification, including removing heavy metal ions, filtrating aggregated particles and reducing dye concentration. The porous media pave the way for practical water desalination and purification.

10:56AM H29.00003 ABSTRACT WITHDRAWN –

11:09AM H29.00004 Ion transport through a charged cylindrical membrane pore contacting stagnant diffusion layers , MATHIAS B. ANDERSEN, Stanford University, P.M. BIESHEUVEL, Wageningen University and Wetsus, MARTIN Z. BAZANT, MIT, ALI MANI, Stanford University — Fundamental understanding of the ion transport in membrane systems by diffusion, electromigration and advection is important in widespread processes such as de-ionization by reverse osmosis and electrodialysis and electro-osmotic micropumps. Here we revisit the classical analysis of a single cylindrical pore, see e.g. Gross and Osterle [J Chem Phys **49**, 228 (1968)]. We extend the analysis by including the well-established concept of contacting stagnant diffusion layers on either side of the pore; thus, the pore is not in direct equilibrium with the reservoirs. Inside the pore the ions are assumed to be in quasi-equilibrium in the radial direction with the surface charge on the pore wall and we obtain a 1D model by area-averaging. We demonstrate that in some extreme limits this model reduces to simpler models studied in the literature; see e.g. Yaroshchuk [J Membrane Sci **396**, 43 (2012)]. Using our model we present predictions of important transport effects such as variation of transport numbers inside the membrane, onset of limiting current, and transient dynamics described by the method of characteristics.

11:22AM H29.00005 Reactive geochemical transport modeling of CO_2 in porous media , MOHAMMAD ALIZADEH NOMELI, AMIR RIAZ, University of Maryland College Park — In this study the modified Redlich-Kwong equation of state is used to develop a pressure-volume-temperature-composition (PVTx) model that predicts how temperature, pressure and salinity affect the solubility of the supercritical CO_2 in brine and is subsequently employed to determine the density and rate of mineral trapping of CO_2 in the form of precipitates. Rates of dissolution and precipitation of minerals are determined by taking into account the pH of the system, in addition to the consideration of the influence of temperature. This study also presents a model to simulate a reactive fluid within permeable porous media. Fluid convection, diffusion and chemical reactions inside a finite space are considered as a simplified representation of natural mineral trapping. The purpose of the current study is to show the time evolution of the aperture shrinkage caused by precipitation of calcite. Precipitation of calcite decreases the porosity and subsequently can change the permeability. Permeability of the porous media controls the path of aqueous CO_2 migration; therefore the aperture width has a pivotal role on solubility and mineral trapping of injected CO_2. The current model predicts the actual efficiency of the mineral trapping mechanism.

11:35AM H29.00006 Relevance of Linear Stability Results to Enhanced Oil Recovery[1] , XUERU DING, PRABIR DARIPA, Texas A&M University — How relevant can the results based on linear stability theory for any problem for that matter be to full scale simulation results? Put it differently, is the optimal design of a system based on linear stability results is optimal or even near optimal for the complex nonlinear system with certain objectives of interest in mind? We will address these issues in the context of enhanced oil recovery by chemical flooding. This will be based on an ongoing work.

[1]Supported by Qatar National Research Fund (a member of the Qatar Foundation).

11:48AM H29.00007 Sustained Reaction Waves Against Flow in Porous Medium: Frozen Fronts , DOMINIQUE SALIN, University Pierre Marie CURIE Paris 6, SEVERINE ATIS, HAROLD AURADOU, CNRS, SANDEEP SAHA, Post Doc, LAURENT TALON, CNRS — Autocatalytic reactions lead to fronts propagating as solitary, self-sustained, waves with a constant velocity and an invariant, flat, concentration profile resulting from a balance between reaction and diffusion. In the presence of a hydrodynamic flow, such fronts, while propagating at a new constant velocity, adapt their shape in order to achieve a balance between reaction diffusion and flow advection all over the front. The issue addressed here is the behaviour of autocatalytic reaction fronts when the forced advection is a heterogeneous flow field. It has been recently observed that in inside a porous medium there exist static, frozen, fronts over a wide range of mean flow rates in the opposite direction of the chemical wave propagation. To account for this dynamical equilibrium where the front is pinned at different points, we use both designed experiments around different configurations of solid obstacles and lattice Boltzmann numerical simulations which allows a control of the flow field heterogeneities. These approach allows us to account for the dependence of the range of observation of frozen states with th control parameters. In the case of the porous medium flow field, the transition to this frozen state is understood in term of percolation like path.

12:01PM H29.00008 The "coffee-ring effect" as a way to remove pollutants and control drying rate in porous media , EMMANUEL KEITA, PAMÉLA FAURE, STÉPHANE RODTS, Universite Paris-Est, Laboratoire Navier, France, DAVID A. WEITZ, Department of Physics, Harvard University, USA, PHILIPPE COUSSOT, Universite Paris-Est, Laboratoire Navier, France — Due to the transport of elements they induce imbibition-drying cycles are known to play a major role in the colloid-facilitated transport in soils and building materials. We study the drying of a colloidal suspension in a porous media. The critical physical phenomenon at work here is the displacement and redistribution of colloidal particles or ions induced by evaporation of the liquid phase from the porous medium. This can be clearly seen by filling a bead packing with coffee. Indeed after full drying the sample has shaded tones with darker regions around the sample free surface and white regions almost free of particles around the bottom. The mechanisms are not yet fully understood and there is no straightforward observation and simple quantification of the spreading of the elements. Using a new MRI technique to look at a complex porous media with colloidal particles in suspension in water we show that the drying of a porous medium filled with elements in the advection regime develops a specific coffee-ring effect. We can quantify how the elements migrate towards the free surface of the sample and accumulate in the remaining liquid films. Our complete understanding of the process makes it possible to establish a simple model predicting the drying rate and the concentration distribution. This opens the way to a control of salt or colloid transport and drying rate of soils and building materials.

Monday, November 19, 2012 10:30AM - 12:27PM –

Session H30 Colloids and Fibers 33A - Chair: Aditya Khair, Carnegie Mellon University

10:30AM H30.00001 Direct Measurements of Colloidal Hydrodynamics near Flat Boundaries , HYUK KYU PAK, CHUNGIL HA, Department of Physics, Pusan National University, DANIEL H. OU-YANG, Department of Physics, Lehigh University, DONG-YUN LEE, Department of Physics, Pusan National University — We studied the hydrodynamic interaction between a colloidal particle close to flat rigid boundaries and the surrounding fluid using oscillating optical tweezers. A colloidal particle located near walls provides a model system to study the behavior of more complex systems whose boundaries can be modeled as effective walls, such as a blood tube, cell membrane, and capillary tube in bio-MEMS. In this study, we measure the hydrodynamic interaction directly without using the Stokes-Einstein relation. Two different cases are studied: a colloidal sphere near a single flat wall and a colloidal sphere located at the midplane between two flat walls. The colloidal hydrodynamics is measured as a function of the distance between the particle and the walls, and is compared with the theoretical results from well-defined hydrodynamics approximations.

10:43AM H30.00002 ABSTRACT WITHDRAWN –

10:56AM H30.00003 Phase transition in non-brownian fiber suspensions[1] , ALEXANDRE FRANCESCHINI, EMMANOUELLA FILIPPIDI, NYU, ELIZABETH GUAZZELLI, Polytech Marseille, France, DAVID PINE, NYU — The simple shear of a suspension of fibers tends to align them with the flow direction. We previously reported that the oscillatory shear of neutrally buoyant non-Brownian fibers align them with the vorticity (Franceschini A. et al. PRL, 2011). We interpreted this phenomenon as the minimization of a "corrected volume fraction" defined as a function of the strain amplitude, the average orientation and the volume fraction. Below a critical value of this parameter, the system becomes fully reversible after a few periods. Above it, fluctuations remain and the fibers align with the vorticity, subsequently reducing the value of this corrected volume fraction. We present here the collective behavior of fibers constrained at the liquid-air interface. By pinning the liquid on the wall of a Couette cell, we can have a flat interface. By modifying the surface of the fibers, we get rid of most of surface tension mediated fiber-fiber interactions. In this 2D configuration we can measure spatial correlations, as well as the position and orientation of every fiber at each shear cycle. We similarly define a "corrected surface fraction" and see how this parameter help us understand the difference between the surface behavior and the suspension behavior.

[1]This work was supported by the NSF through the NYU MRSEC, Award DMR:0820341. Additional support was provided by a Lavoisier Fellowship (AF) and from the Onassis Foundation (EF)

11:09AM H30.00004 Rotational motion of a thin axisymmetric disk in a low Reynolds number linear flow , VIKRAM SINGH, DONALD KOCH, Cornell University, GANESH SUBRAMANIAN, Jawaharlal Nehru Centre for Advanced Scientific Research, ABRAHAM STROOCK, Cornell University — The problem of a single particle motion at low Reynolds is one of the most fundamental problems of fluid mechanics. It is rather surprising that despite our deep understanding of particle motion at large aspect ratio we know very little about particles other than spheroids at small aspect ratio. In this work, motion of thin axisymmetric rigid particles with fore-aft symmetry in simple shear flow is investigated. We determine the scaling of the effective aspect ratio for a family of shapes given by, $y(\rho) = \kappa(1-\rho^2)^\alpha$ where α is a positive parameter, ρ is the radial distance in polar coordinates, $y$is the thickness of the particle, κ is the aspect ratio of the particle, and effective aspect ratio is defined as the aspect ratio of a spheroid having the same period of rotation as that of the particle. For an axisymmetric particle, effective aspect ratio can be determined based on the torques acting on the particle in two different orientations. Starting with the integral representation of Stokes flow, matched asymptotic analysis is performed to determine the scaling of the torque acting on a stationary particle in simple shear flow with κ as the small parameter. Using boundary element method simulations, the exact torques are obtained and the scaling of effective aspect ratio obtained from the analysis is verified.

11:22AM H30.00005 Vorticity alignment of rigid fibers in oscillatory flow , JASON BUTLER, BRADEN SNOOK, Dept. of Chem. Engr., University of Florida, DEPT. OF CHEM. ENGR., UNIVERSITY OF FLORIDA TEAM — Rigid fibers suspended at high concentration in a viscous, Newtonian fluid can be aligned perpendicular to the flow-gradient plane by applying an oscillatory shear flow. Direct comparisons with published experiments of Franceschini and coworkers [Phys. Rev. Letters 107, 250603 (2011)] demonstrate that a simple model, which considers only excluded volume and self-mobilities, can accurately predict the orientation distributions. However simulations reveal that this surprising alignment occurs only if the ratio of the gap width to fiber length is small (i.e. a highly confined suspension). Stresses calculated from the numerical simulations are also reported and compared to the experimentally measured rheology.

11:35AM H30.00006 The onset of particle-dominated convection regime in colloidal suspensions , LAYACHI HADJI, MAHMOUD DARASSI, The University of Alabama — In the experiments of Bénard convection in a suspension of microparticle by Chang *et al.* (2008), a parameter β was isolated to model the interplay between the effects of thermophoresis, sedimentation and Brownian diffusion. A plot of β as function of the particle radius, r_p, for a suspension of aluminum oxide particles in water shows that the function $\beta(r_p)$ has the shape of an inverted parabola with two roots so that $0 < \beta \ll 1$ for 1 nm $\leq r_p \leq 5$ nm and for $r_p \approx 50$ nm where thermophoresis and sedimentation are balanced. We consider a particulate medium model to determine the threshold instability conditions. Due to the large particle size, the convection process is characterized by longer diffusion time scale, much smaller Lewis number, τ, and larger separation ratio, S, than the binary mixture case. For $0 < \beta \ll 1$ which corresponds to two distinct particle radii, a small wavenumber expansion yields the value $R_c = 720\,\tau/S$. For $\beta = O\big(1\big)$, threshold stability conditions are depicted as function of the particle size and the height of the fluid cell.

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

11:48AM H30.00007 The influence of frequency dependent impedance properties on electrohydrodynamic aggregation of colloidal particles , T.J. WOEHL, C.S. DUTCHER, N.H. TALKEN, W.D. RISTENPART, Dept. Chemical Engineering and Materials Science, Univ. California Davis — Colloidal particles suspended in dilute electrolytes have been widely observed to aggregate laterally along electrodes in response to oscillatory electric fields, a phenomenon that has been generally attributed to electrohydrodynamic fluid flow. Fundamental aspects of the aggregation behavior, however, remain unclear. Recently, our group has observed a second order transition in the order parameter for colloidal aggregates for a variety of electrolytes over the frequency range of 100 to 500 Hz. Here we explore the frequency dependence on several parameters related to the electrochemical cell impedance, including the AC current density, an apparent DC current offset, and the current-voltage phase angle. We report the AC current density increases by at least 10% from 100 Hz to 500 Hz for both NaCl and KCl aqueous solutions, a range commensurate with the observed second order transition. We investigate how the frequency dependent impedance properties affect the aggregation of colloidal particles and discuss the implications for controlling the crystallinity of the colloidal aggregates.

12:01PM H30.00008 Richardson Dispersion in Brownian Motion , EMMANUEL VILLERMAUX, Aix Marseille University, JEROME DUPLAT, Universite Joseph Fourier — Since Langevin, the Brownian motion of a microscopic particle explicitly accounts for a short-time correlated "thermal" force. The motion is ballistic, $\langle x^2\rangle \sim t^2$ at short time scales, and diffusive $\langle x^2\rangle \sim t$ at long time scales, where x is the displacement of the particle during time t, and the average is taken over the thermal distribution of initial conditions. High Reynolds number turbulence is known to exhibit a régime called Richardson dispersion, in which the *relative* separation δx of material particles grows super-diffusively. Namely, $\langle \delta x^2\rangle \sim t^3$, with the average taken over many particles released from the same initial conditions. We show that Richardson dispersion is indeed property of Brownian motion, under the condition that the initial velocity is fixed rather than distributed thermally. We analyze the motion of an optically trapped particle in air, and indeed find t^3 dispersion. This super-diffusive régime, unveiled here, is the direct proof of the existence of the random, rapidly varying force imagined by Langevin, and reveals a profound similarity between molecular diffusion at microscopic scales and turbulent diffusion at much larger scales.

12:14PM H30.00009 Advective symmetry breaking in phoretic motion of colloidal particles , ADITYA KHAIR, Department of Chemical Engineeering, Carnegie Mellon University — The phoretic motion of a colloidal particle is animated by an imposed gradient of a scalar field: e.g. solute gradients cause diffusiophoresis and temperature gradients drive thermophoresis. It is customarily assumed that the scalar field evolves solely via diffusion (i.e. the Peclet number is zero). This leads to Morrison's remarkable result that the translational phoretic velocity of a colloid is independent of its size, shape, and orientation relative to the imposed gradient (the colloid does not rotate, either). Moreover, colloids comprising a dispersion are predicted to translate with identical velocities. However, intuitively, as a colloid moves it sets up a fluid flow that advects the same scalar field that instigated its motion. Here, using asymptotic analysis, we explore the first effects of advection on the phoretic motion of colloidal particles (i.e. at the experimentally relevant conditions of small but finite Peclet number). We show that advection leads to symmetry breaking in the phoretic motion of fore-aft asymmetric particles, where the particle velocity depends on the direction of the imposed gradient. We demonstrate that advection drives phoretic rotation of nonspherical colloids. Last, advection is shown to cause relative motion between colloidal particles.

Monday, November 19, 2012 10:30AM - 12:27PM –

Session H31 Particle Laden Flows IV 33B - Chair: Said Elghobashi, University of California, Irvine

10:30AM H31.00001 Near-surface sea spray dynamics via simulations of particle-laden, turbulent Couette flow , DAVID RICHTER, PETER SULLIVAN, National Center for Atmospheric Research — In the atmospheric surface layer situated over the air-sea interface, high winds can eject large amounts of sea spray into the turbulent flow above. The question of whether or not this dispersed phase within the turbulent surface layer can alter momentum transfer from the air to the ocean surface remains unresolved. This study, therefore, aims to identify and explain modifications of wall-normal momentum transfer in a turbulent, particle-laden flow. This is done using direct numerical simulation (DNS) with a Lagrangian point-particle representation of the dispersed phase. Turbulent Couette flow, chosen since it exhibits certain features similar to the atmospheric surface layer, is investigated with varying concentrations and sizes of spherical, non-interacting particles. Generally speaking, the addition of a dispersed phase disrupts the motions responsible for turbulent, carrier-phase momentum transfer, while at the same time compensating for this loss of momentum transfer through an additional dispersed phase stress. Mechanisms and interpretations of these changes in turbulent wall-normal momentum transport will be presented.

10:43AM H31.00002 Numerical study of the boundary conditions 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. In this presentation, the units transfer between the lattice Boltzmann system and the physical system is characterized in detail, in order to simulate flows from the realistic physical world. Force evaluations, based on the momentum exchange method and the FH model used to implement boundary conditions, are shown for both a static and moving cylinder in a 2D channel. Finally, the settling trajectory of the cylinder after it is released away from the centerline in a Poiseuille flow is investigated with varying Reynolds numbers.

10:56AM H31.00003 DNS of particle dispersion in a spatially developing turbulent boundary layer , MICHAEL DODD, KEEGAN WEBSTER, ANTONINO FERRANTE, University of Washington, Seattle — We performed DNS of particle-laden spatially developing turbulent boundary layer at $\mathrm{Re}_\theta = 1000 - 3200$. We computed the Lagrangian trajectories of millions of fluid points and solid particles of three different Stokes number, St=0.1, 1, and 5. The particles were gradually released from a line source. We computed the time development of particle mean displacement, dispersion, and turbulent diffusivity. Our DNS results of fluid point mean-displacements are in excellent agreement with those of Batchelor's (1964) theory. Also, our DNS results show that in general particle statistics are strongly influenced by particle's Stokes number. Such dependence is mostly caused by the particles tendency to preferentially accumulate in the viscous sublayer as their Stokes number increases. Furthermore, for $t/T_L < 1$ where T_L is the Lagrangian integral time scale, the streamwise and wall-normal dispersions are $\propto t^2$ for fluid points and $\propto t^3$ for solid particles. For $20 < t/T_L < 80$, the streamwise dispersion of fluid points and particles with $\mathrm{St} = 0.1$ is approximately $\propto t^{5/3}$, while that of particles with $\mathrm{St} = 1$ and 5 is approximately $\propto t^{5/2}$. For all cases studied and for $20 < t/T_L < 80$, the wall-normal dispersion is approximately $\propto t$.

11:09AM H31.00004 Modeling near-wall interphase exchanges for particle-laden flows , OLIVIER DESJARDINS, JESSE CAPECELATRO, Cornell University, NATIONAL RENEWABLE ENERGY LAB COLLABORATION — In Eulerian-Lagrangian and Eulerian-Eulerian modeling approaches of dispersed multiphase flows, proper treatment of mass and momentum transfer between the phases is required to capture the correct physical behavior. Coupling often involves the volume fraction and momentum exchange term based on correlations for drag. The accuracy of these terms diminishes at regions close to walls, where key assumptions that were used in the formulation of the models are often violated. Defining particle volume fraction close to a solid boundary could require using detailed information on the distance between the surface of the particles and the wall. No-slip boundary conditions are imposed on the fluid phase while particles may slip, complicating the momentum transfer. In addition, experiments have reported enhanced lift at the walls, corresponding to values greater than what can be estimated from Saffman shear-induced models. In this study, coupling between the phases is handled in an Euler-Lagrange framework using a two-step filtering process that ensures a conservative exchange, as well as convergence under mesh refinement. A turbulent spout fluidized bed is simulated, and compared to experimental data. Different strategies are explored to properly account for the presence of the walls.

11:22AM H31.00005 Particle equilibrium in 3D-channel flow for one and two particles , JOY KLINKENBERG, Technische Universiteit Eindhoven & KTH Stockholm, H.C. DE LANGE, Technische Universiteit Eindhoven, WIM-PAUL BREUGEM, Technische Universiteit Delft, LUCA BRANDT, KTH Stockholm — We perform Direct Numerical Simulations of an Euler-Lagrange coupled particle-laden channel flow to investigate the equilibrium position of particles. The channel is periodic in both stream- and spanwise direction, with no-slip on top and bottom walls. Particles are neutrally buoyant and modeled using the Immersed Boundary Method, with a dimater of 20% of the channel height. The lateral movement of one and two particles is studied for Reynolds numbers, based on channel height and bulk velocity, between 5 and 1000. We show that with a Reynolds number change, the equilibrium position changes. To investigate the effect of periodicity, several channel lengths and widths have been investigated. Also the effect of particle-particle interaction on the equilibrium is investigated by modelling 2 particles, influencing each other.

11:35AM H31.00006 Particle Dynamics in Rotating Flow inside Coaxial Cylinder , ALBERT S. KIM, University of Hawaii at Manoa, SUNGSU LEE, Chungbuk National University — In this study, trajectory and distribution of unequal-sized particles in a coaxial cylinder are investigated using dissipative hydrodynamics (DHD), an updated version of Stokesian dynamics. Flow field is established by rotating an inner cylinder in a fixed outer cylinder. Initially, particles are randomly released in the flow. The flow field is then decomposed into unidirectional flow, vortice and rate-of-strain at particle centers. Translation and rotation of particles are accurately mimicked using the fourth-rank hydrodynamic tensors. In general, far-field many-body grand mobility matrix was formed at each time step as a function of particle positions, and inverted to calculate the grand resistance matrix. The Langevin equation is directly solved to trace the particle trajectory using the intermediate time step to physically mimic influence of force and torque. Particle inertia is intrinsically included as rotational fluid speed increases from the surface of the inner cylinder to that of the outer cylinder. Optimal operation conditions to separate particles due to size differences are suggested by DHD simulation results. This work was financially supported by projects of the "Development of Energy utilization technology with Deep Ocean Water," KIOST of Korea.

11:48AM H31.00007 Numerical studies of the effects of neutrally buoyant large particles on turbulent channel flow at the friction Reynolds number up to 395[1] , ZHAOSHENG YU, YU WANG, XUEMING SHAO, State Key Laboratory of Fluid Power Transmission and Control, Department of Mechanics, Zhejiang University — A direct-forcing fictitious domain method was employed to perform fully-resolved numerical simulations of turbulent channel flow laden with large neutrally buoyant particles at constant pressure gradients. The effects of the particles on the turbulence (including the fluid-phase average velocity, the root-mean-square (rms) of the velocity fluctuation, the probability density function of the velocity and the vortex structures) at the friction Reynolds number of 180 and 395 were investigated. The results show that the drag-reduction effect caused by the spherical particle at low particle volumes is very small. The presence of particles decreases the maximum rms of streamwise velocity fluctuation near wall via weakening the large-scale streamwise vortices, and on the other hand increases the rms of transverse and spanwise fluctuating velocities in vicinity of the wall via inducing smaller-scale vortices. The effects of the particles on the fluid velocity PDF (probability density function) normalized with the rms velocity are small, irrespective of the particle size, particle volume fraction and Reynolds number.

[1]The work was supported by the National Natural Science Foundation of China (Nos. 11072217 and 11132008), the Fundamental Research Funds for the Central Universities, and the Program for New Century Excellent Talents in University.

12:01PM H31.00008 Four-way coupling simulation of particle-laden turbulent channel flow , JUNGHOON LEE, CHANGHOON LEE, Yonsei University — Transport of small inertial particles near a wall in turbulent flows is frequently observed in various engineering applications. In this kind of flow, the gas-phase turbulence level may be modified due to the presence of the particles. Furthermore, for sufficiently high volume fractions, particle-particle interactions strongly influence particle dispersion and thus four-way coupling becomes essential in simulation of particle-laden turbulence. In this study, we investigate inertial particle motion in near-wall turbulence using direct numerical simulations. The effects of inter-particle collisions and particle feedback on the fluid were taken into account in our spectral simulation. It is assumed that inertial particle motion is governed by Stokes drag, lift and gravitationa forces. Particle deposion, velocity and acceleration statistics are discussed for various Stokes numbers and particle volume fractions. The particle Stokes number is defined as the particle response time normalized by the wall units. The Stokes number range considered is 25$\sim$2,000 and particle volume fraction ranges from 3×10^{-6} to 6×10^{-5}. We also compare our numerical results with available experimental measurements. Detailed results will be presented in the meeting.

12:14PM H31.00009 Behavior of particles in turbulence over a wavy wall , HEA EUN LEE, CHANGHOON LEE, Yonsei University — Particle motion in near-wall turbulence plays an important role in many physical processes such as sediment transport and pollution control. There have been many studies which focused on particles in turbulence over a flat wall. Behavior of particles over a rough wall, however, was not investigated much. In this study, particle motion in turbulent flow over a wavy wall is investigated using direct numerical simulation. The wave-induced variation of flow is simulated by spectral method and compared with the flow over a flat wall. The virtual boundary method proposed by Goldstein et al. (1995) is applied to impose no-slip condition at wavy boundary. To begin with, we focused on the differences between turbulence generated at a wavy boundary and one at a flat wall such as friction factors, velocity fluctuations, and vortical structures associated with shear layers that form behind the wave. Also, focusing on the mechanism controlling the inertial particles in turbulence, particle motion in turbulence over wavy wall is investigated. Due to the turbulent structure modified by wavy geometry, inertial particles are clustering in upslope part of the wall which is the region with high shear stress. Detailed particle statistics over a wavy wall will be discussed in the meeting.

Monday, November 19, 2012 10:30AM - 12:14PM –

Session H32 Granular Flows II 33C - Chair: Yang Ding, Georgia Institute of Technology

10:30AM H32.00001 Drag and lift forces in granular flows , FRANÇOIS GUILLARD, OLIVIER POULIQUEN, YOËL FORTERRE, Aix-Marseille Universite, CNRS — Forces exerted on obstacles moving in a granular medium are studied both experimentally and numerically. The experiment consists in an horizontal cylinder rotating around a vertical axis in a granular media. Both drag forces and lift forces experienced by the cylinder are measured. The first striking result is obtained during the first half rotation, before the cylinder crosses its wake. Despite the symmetry of the object, a strong lift force is measured, about 20 times the buoyancy. The scaling of this force is studied experimentally and discussed in the framework of a continuum model based on a frictional rheology. The second striking observation is made after several rotations. The drag force dramatically drops and becomes independent of depth, showing that it no longer scales with the hydrostatic pressure. The rotation of the cylinder induces a structure in the packing, which screens the weight of the grains above.

10:43AM H32.00002 From antinode clusters to node clusters: The concentration dependent transition of floaters on a standing Faraday wave[1] , CEYDA SANLI, Okinawa Institute of Science and Technology, DETLEF LOHSE, DEVARAJ VAN DER MEER, Physics of Fluids, University of Twente — A hydrophilic floating sphere that is denser than water drifts to an amplitude maximum (antinode) of a surface standing wave. A few identical floaters therefore organize into antinode clusters. However, beyond a transitional value of the floater concentration ϕ, we observe that the same spheres spontaneously accumulate at the nodal lines, completely inverting the self-organized particle pattern on the wave. From a potential energy estimate we show that at low ϕ antinode clusters are energetically favorable over nodal ones and how this situation reverses at high ϕ, in agreement with experiment.

[1]The work is part of the research program of FOM, which is financially supported by NWO; C.S. acknowledges financial support.

10:56AM H32.00003 Force and flow response of granular matter to simultaneous intruders , PAUL UMBANHOWAR, Northwestern University, LIONEL LONDON, DANIEL GOLDMAN, Georgia Institute of Technology — For two horizontal, parallel rods vertically penetrating into dense granular matter at constant velocity, we examine the grain flow and the vertical and horizontal components of the force as functions of rod separation s using experiment and DEM simulation. In previous experiments we found that while the vertical force F required to maintain constant velocity increases nearly linearly with the penetration depth d, the dependence of the slope F/d on s is more interesting. As s is increased from zero, F/d increases to a maximum value and then slowly decreases to twice the value of F/d for a single rod at large s. Here we present new results examining the relation between the horizontal and vertical forces acting on each rod and the associated granular flow as functions of s, particle friction coefficient μ, and horizontal rod support compliance. Increasing μ increases the value of s at which $F(s)/d$ is maximum, while decreasing compliance reduces the peak in penetration resistance while increasing the range of s where penetration resistance is enhanced. The flow fields around each rod become more asymmetric in the horizontal as s is decreased and exhibit correlations with both the mean and fluctuating components of the forces on the rods.

11:09AM H32.00004 ABSTRACT WITHDRAWN –

11:22AM H32.00005 Force measurements on an intruder in pre-fluidized sand , TESS A.M. HOMAN, DETLEF LOHSE, DEVARAJ VAN DER MEER, Physics of Fluids, University of Twente, Enschede — A container filled with sand is fluidized from below, and by slowly turning of the air flow a very loose packing is created. An intruder controlled by a linear motor is connected to a load cell to measure the force it experiences while moving through the bed. By varying several parameters, such as the intruder shape, velocity, and penetration depth, we aim to obtain a better understanding of the behavior of loosely packed granular materials.

11:35AM H32.00006 A resistive force model for complex intrusion in granular media , TINGNAN ZHANG, School of Physics, Georgia Institute of Technology, CHEN LI[1], Department of Integrative Biology, University of California, Berkeley, DANIEL GOLDMAN, School of Physics, Georgia Institute of Technology — Intrusion forces in granular media (GM) are best understood for simple shapes (like disks and rods) undergoing vertical penetration and horizontal drag. Inspired by a resistive force theory for sand-swimming [1], we develop a new two-dimensional resistive force model for intruders of arbitrary shape and intrusion path into GM in the vertical plane. We divide an intruder of complex geometry into small segments and approximate segmental forces by measuring forces on small flat plates in experiments. Both lift and drag forces on the plates are proportional to penetration depth, and depend sensitively on the angle of attack and the direction of motion. Summation of segmental forces over the intruder predicts the net forces on a c-leg, a flat leg, and a reversed c-leg rotated into GM about a fixed axle. The stress profiles are similar for GM of different particle sizes, densities, coefficients of friction, and volume fractions. We propose a universal scaling law applicable to all tested GM. By combining the new force model with a multi-body simulator, we can also predict the locomotion dynamics of a small legged robot on GM. Our force laws can provide a strict test of hydrodynamic-like approaches to model dense granular flows.

[1] Maladen et al, J. R. Soc. Interface, 8, 1332, (2011)

[1]Also affiliated to: School of Physics, Georgia Institute of Technology

11:48AM H32.00007 Sidewinding snakes on sand[1] , HAMIDREZA MARVI, DANTE DIMENICHI, ROBERT CHRYSTAL, Georgia Institute of Technology, JOSEPH MENDELSON, Zoo Atlanta and Georgia Institute of Technology, DANIEL GOLDMAN, DAVID HU, Georgia Institute of Technology, GEORGIA TECH AND ZOO ATLANTA COLLABORATION — Desert snakes such as the rattlesnake *Crotalus cerastes* propel themselves over sand using sidewinding, a mode of locomotion relying upon helical traveling waves. While sidewinding on hard ground has been described, the mechanics of movement on more natural substrates such as granular media remain poorly understood. In this experimental study, we use 3-D high speed video to characterize the motion of a sidewinder rattlesnake as it moves on a granular bed. We study the movement both on natural desert sand and in an air-fluidized bed trackway which we use to challenge the animal on different compactions of granular media. Particular attention is paid to rationalizing the snake's thrust on this media using friction and normal forces on the piles of sand created by the snake's body.

[1]The authors thank the NSF (PHY-0848894), Georgia Tech, and the Elizabeth Smithgall Watts endowment for support. We would also like to thank Zoo Atlanta staff for their generous help with this project.

12:01PM H32.00008 Lift-off performance of a jumping robot on hard and soft ground , JEFFREY AGUILAR, ALEX LESOV, KURT WIESENFELD, DANIEL GOLDMAN, Georgia Tech — We study lift-off during jumping on hard ground and granular media in a simple robot composed of a linear actuator in series with a spring. On hard ground, the robot jumps from a metallic base. On granular media (GM) composed of 0.3 mm glass particles, a circular foot is attached to the spring and an air-fluidized bed sets the initial volume fraction, ϕ. The actuator frequency and phase are systematically varied to find optimal performance. On both substrates, optimal jump height does not occur at the robot's resonant frequency f_0. Two distinct jumping modes emerge: a simple jump which is optimal above f_0 is achievable with a squat maneuver, and a "stutter" jump which is optimal below f_0 is generated with a counter-movement. For hard ground, both modes exhibit similar performance. On closely packed GM ($\phi = 0.62$), the simple jump becomes the favored mode. On loosely packed GM ($\phi = 0.58$), jump height performance is significantly reduced due to greater yielding in the material. A dynamical model reveals how optimal lift-off results from non-resonant transient dynamics.

Monday, November 19, 2012 10:30AM - 12:27PM –

Session H33 Mini-Symposium: Complex Fluid Flows in Memory of Daniel D. Joseph II 29A - Chair: Howard Hu, University of Pennsylvania

10:30AM H33.00001 Linear Instabilities in Simple Shear Flow of Polymer Solutions Driven by Stress-gradient/Concentration Coupling , GARY LEAL, MICHAEL CROMER, MICHAEL VILLET, GLENN FREDRICKSON, University of California, Santa Barbara — Inhomogeneities in the flow of polymer solutions have been observed in experiment over the last 40 years, e.g. including the well studied formation of shear bands in a linear shear flow device. In this study we investigate the role of concentration non-uniformities in causing hydrodynamic instabilities in a linear shear flow. To incorporate a stress-concentration coupling we follow a two-fluid model formalism, in which the polymer conformation is described by the Rolie-Poly model. In a shear flow, the model may exhibit linear instability due to perturbations in the gradient direction, even for a monotonic constitutive curve, provided there is a sufficient separation between the characteristic polymer relaxation times. We show that the mechanism driving the instability is the motion of polymer up stress gradients, leading to concentration nonuniformities in the flow.

10:43AM H33.00002 Brownian Swimming via Taylor Dispersion , JOE GODDARD, ERIC LAUGA, University of California, San Diego — We show that the theory of generalized Taylor dispersion can be employed to analyze a model of a low-Re swimmer undergoing Brownian tumbling coupled with systematic translation along a preferred axis. The resulting formula for translational diffusivity confirms a previous analysis[1] based on Langevin dynamics. This present approach may provide a useful method for treating more complex stochastic swimmers.

[1]Lauga, E., PRL 106, 178101 (2011)

10:56AM H33.00003 Active Nematic Flows , GREG FOREST, UNC Chapel Hill, QI WANG, University of South Carolina, RUHAI ZHOU, Old Dominion University — The recent flurry of activity in swimming particle suspensions is extended to macromolecular rods by incorporating polarity, active stress, and density gradients into the kinetic theory of nematic polymers. Simulations predict phenomena unique to nano-rod swimmers at dilute and semi-dilute concentrations.

11:09AM H33.00004 Apparent viscosity during unyielding of a thixotropic yield stress fluid[1] , YURIKO RENARDY, Dept of Mathematics, 460 McBryde Hall, Virginia Tech, Blacksburg, VA 24061-0123, KARA MAKI, School of Mathematical Sciences, Rochester Institute of Technology, Rochester, NY 14623 — We present a mathematical interpretation of a thixotropic yield stress fluid, based on a viscoelastic constitutive law in the limit of large relaxation time, together with a Newtonian solvent. The dynamics is initiated by a step-up or step-down in prescribed shear stress. There is no presumption of a yield stress, but nevertheless, we obtain yield stress behavior. The thixotropic behavior of the model arises from the multiple time scales which emerge in the limit of large relaxation time. These give rise to fast dynamics (elastic deformation) and slow dynamics (unyielding), in addition to yielded dynamics for shear flow. We present how the model predicts the evolution of apparent viscosity during unyielding.

[1]Supported by NSF-DMS, AWM.

11:22AM H33.00005 A universal constraint-based formulation for freely moving immersed bodies in fluids[1] , NEELESH A. PATANKAR, Northwestern University — Numerical simulation of moving immersed bodies in fluids is now practiced routinely. A variety of variants of these approaches have been published, most of which rely on using a background mesh for the fluid equations and tracking the body using Lagrangian points. In this talk, generalized constraint-based governing equations will be presented that provide a unified framework for various immersed body techniques. The key idea that is common to these methods is to assume that the entire fluid-body domain is a "fluid" and then to constrain the body domain to move in accordance with its governing equations. The immersed body can be rigid or deforming. The governing equations are developed so that they are independent of the nature of temporal or spatial discretization schemes. Specific choices of time stepping and spatial discretization then lead to techniques developed in prior literature ranging from freely moving rigid to elastic self-propelling bodies. To simulate Brownian systems, thermal fluctuations can be included in the fluid equations via additional random stress terms. Solving the fluctuating hydrodynamic equations coupled with the immersed body results in the Brownian motion of that body. The constraint-based formulation leads to fractional time stepping algorithms a la Chorin-type schemes that are suitable for fast computations of rigid or self-propelling bodies whose deformation kinematics are known.

[1]Support from NSF is gratefully acknowledged.

11:35AM H33.00006 Computationally and experimentally assessed base-flow, stability, and sensitivity differences between shear dominated (negligible gravity) and gravity assisted internal condensing flows[1] , AMITABH NARAIN, RANJEETH NAIK, SOUMYA MITRA, MICHAEL KIVISALU, Mechanical Engineering, Michigan Technological University, Houghton, MI-49931 — Annular regimes for internal condensing flow are desirable for high heat transfer rates out of a condenser. Predominantly shear driven flows typically occur in horizontal channels (with condensation on the bottom horizontal-surface), zero gravity flows, and in millimeter to micro-meter scale hydraulic diameter ducts. This talk presents steady and unsteady computational results obtained from the numerical solutions of the full two-dimensional governing equations for annular internal condensing flows in a channel. Results obtained for inclined, horizontal, and zero-gravity cases (with and without surface-tension) bring out the differences between shear driven and gravity assisted/driven flows. The results highlight the differences between steady solutions, their stability, and their noise-sensitivity. It is shown that annular flows are more stable and easily realized for gravity driven or gravity assisted flows than for primarily shear driven flows. Besides stability, extreme-sensitivity of shear driven flows to typically present persistent fluctuations is also demonstrated. This sensitivity is beneficially exploited to achieve significant heat-transfer rate enhancements. The talk also highlights conditions for which surface tension forces become important. The computational results have been validated by good comparisons with condensing flow experimental results for the annular regimes.

[1]NSF-CBET-1033591

11:48AM H33.00007 A New Model for Instantaneous Coal and Gas Outbursts , KANGPING CHEN, Arizona State University — An instantaneous coal and gas outburst is a sudden and violent simultaneous ejection of large amounts of coal and gas from the working coalface during underground mining. Existing theories are incapable of explaining many precursors of an outburst, which include the occurrence of distinct audible noises originating close to the mine opening and a decrease in the temperature in the solid of the coalface and the nearby atmosphere. Nor can they explain the increased proneness to outbursting with increased rate-of-advance of the coalface. They are incapable of predicting a failure of explosive and catastrophic nature which characterizes an instantaneous outburst. A new model combining fracture mechanics, gas dynamics and rock mechanics is presented to elucidate the physical mechanisms leading to instantaneous coal and gas outbursts. This model suggests a domino effect that causes a catastrophic failure of the coal and an instantaneous outburst; it identifies a critical condition for the onset of an outburst, and it successfully predicts all of the observed phenomena preceding outbursts. The model also predicts a fracture aperture size effect which is confirmed by observations.

12:01PM H33.00008 Transient Flow due to the Adsorption of Particles , PUSHPENDRA SINGH, NAGA MUSUNURI, BHAVIN DALAL, IAN FISCHER, DANIEL CODJOE, NJIT — When small particles, e.g., glass, flour, pollen, etc., come in contact with a fluid-liquid interface they disperse so quickly to form a monolayer on the surface that it appears explosive, especially on the surface of mobile liquids like water. This is a consequence of the fact that a particle coming in contact with a liquid surface is pulled into the surface by capillary force causing the particle to accelerate to a relatively-large velocity in the direction normal to the surface. This vertical motion of the particle gives rise to a lateral flow on the surface away from the particle. PIV measurements show that the adsorption of a spherical particle causes a transient axisymmetric flow about the vertical line passing through the center of the particle. The flow develops in a fraction of second after the adsorption of the particle and persists for several seconds. The fluid directly below the particle rises upwards and near the surface it moves away from the particle.

12:14PM H33.00009 Thermal Dielectrophoretic (T-DEP) Force , HOWARD HU, BARUKYAH SHARARENKO, HAIM BAU, University of Pennsylvania — When subjected to a non-uniform electric field, a dielectric particle in a dielectric medium experiences a dielectrophoretic (DEP) force. For some applications in microfluidic systems, thermal effects due to Joule heating are quite important. In this study, we examine the additional dielectrophoretic force due to the thermal effect, which we termed as thermal dielectrophoretic (T-DEP) force. A thermal gradient may be established in the fluid due to Joule heating, which leads to the spatial variations in conductivity and permittivity of the fluid. With the gradients in the conductivity and permittivity, an electric field (even a uniform field) will induce electric forces in the fluid (and in the particle), and cause a flow (electrothermal flow). We have derived an expression for the net thermal dielectrophoretic (T-DEP) force acting on a particle suspended in a medium with a temperature gradient. This extra T-DEP force has never been discussed in literature, could be important in predicting the particle trajectories in such flow systems, and explain the discrepancy observed between the theoretical prediction and experimental measurements.

Monday, November 19, 2012 2:00PM - 2:35PM –

Session J33 Invited Session: Boundary Layers in Favourable Pressure Gradients Ballroom 20A - Chair: Sutanu Sarkar, University of California, San Diego

2:00PM J33.00001 Boundary layers in favourable pressure gradients , UGO PIOMELLI, Queen's University, Kingston (ON) — Turbulent boundary layers subjected to freestream acceleration due to a favorable pressure gradient (FPG) are common in many engineering applications. For strong acceleration the flow tends to revert to a laminar state; whether it re-laminarizes fully depends on the strength of the acceleration, and on the distance over which the acceleration is maintained. As the pressure gradient is removed, the flow may then return to a turbulent state; the re-transitioning process is strongly affected by the state of the turbulence at the end of the acceleration region. In this talk we present results of simulations of turbulent flow in flat-plate boundary layers subjected to strong acceleration, exceeding the critical Reynolds number for extended distance. Two Reynolds numbers are considered: a low one is studied by direct simulations, a higher one by large-eddy simulations. As the acceleration increases, the logarithmic layer is initially preserved, albeit with a higher value of the von Kàrmàn constant; in the region of high acceleration, however, the velocity profile becomes laminar-like; in the high-Re case, a new logarithmic layer is established shortly after the end of the acceleration, while in the low-Re case re-transition occurs much later. Good agreement of the high-Re LES with the experimental data is observed. The region of maximum acceleration is characterized by significant reorganization of the wall layer, with streaks that remain stable for very long distances. Frozen turbulence advected from upstream is still present, but it does not adjust to the freestream acceleration (i.e., the freestream velocity increases, but the turbulent kinetic energy maintains its upstream value); the residual turbulent fluctuations are large enough that, once the acceleration ends, a bypass-like transition process is triggered.

Monday, November 19, 2012 2:00PM - 2:35PM –

Session J34 Invited Session: Control of Oscillator and Amplifier Flows Ballroom 20BC - Chair: Tom Bewley, University of California, San Diego

2:00PM J34.00001 Control of oscillator and amplifier flows , PETER SCHMID, LadHyX, Ecole Polytechnique — Flow control aims at the targeted manipulation of inherent flow behavior and is a critical component in efforts to delay instabilities, reduce drag, decrease receptivities or extend the operational parameter range of a fluid device. The design of flow control strategies relies on a model for the fluid system but also a model for the noise environment. For flows that are insensitive to external noise (oscillator flows), effective control strategies have been designed with considerable success; for flows that respond sensitively to environmental noise (amplifier flows), however, the design of effective control schemes is far more challenging, as it crucially depends on the quality of the noise model. We will present and discuss the critical steps in the design of flow control schemes for both types of flow behavior and compare and contrast a model based and data-based approach. This presentation summarizes joint work with Denis Sipp (ONERA-DAFE) and various doctoral students.

Monday, November 19, 2012 2:40PM - 3:15PM –

Session K33 Invited Session: Numerical Investigations of Turbidity Currents Ballroom 20A - Chair: Juan Lasheras, University of California, San Diego

2:40PM K33.00001 Numerical Investigations of Turbidity Currents , ECKART MEIBURG, UC Santa Barbara — Turbidity currents are particle-laden, geophysical flows driven by gravity. Within the global sediment cycle, they represent the primary mechanism by which sediment is transported from the continental shelves into the deep ocean, with transport distances ranging up to O(1,000) km. They furthermore influence the formation of an important class of hydrocarbon reservoirs. As turbidity currents propagate along the sea floor, they can trigger the formation of a variety of topographical features through the processes of deposition and erosion, such as channels, levees and sediment waves. The talk will review high-resolution, two- and three-dimensional Navier-Stokes simulations of turbidity currents, along with related linear instability mechanisms, that have advanced our understanding of these phenomena. For the mathematical description, we assume that the particles have negligible inertia and are much smaller than the smallest length scales of the buoyancy-induced fluid motion. Under these conditions, we can employ an Eulerian approach for the description of the particulate phase. Results will be shown regarding the unsteady interaction of turbidity currents with channels, pipelines and local seamounts. We will also address currents in stratified ambient, as well as reversing buoyancy currents.

Monday, November 19, 2012 2:40PM - 3:15PM –

Session K34 Invited Session: Tipstreaming and other methods of producing fine fluid threads Ballroom 20BC - Chair: Geno Pawlak, University of California, San Diego

2:40PM K34.00001 Tipstreaming and other methods of producing fine fluid threads , SHELLEY ANNA, Carnegie Mellon University — Capillary breakup has long been used to generate uniform fluid droplets, but capillary breakup alone does not easily yield drop sizes below the micrometer scale. Additional factors can be combined to effectively promote uniform small-scale breakup. A well-established example is the use of piezoelectric forcing, such as in inkjet printing. More recent approaches to forming micron and submicron droplets include microfluidic methods using confined geometries, use of interfacial tension gradients including tipstreaming to produce fine threads, and electrosprays and electrohydrodynamic jetting. These methods aid in the generation of fine fluid threads that subsequently fragment due to capillarity. In this talk I will compare these methods of producing uniform micron and submicron droplets, and I will discuss in particular recent work to describe and control surfactant-mediated tipstreaming in microfluidic devices.

Monday, November 19, 2012 3:35PM - 5:45PM –

Session L1 Geophysical: Atmospheric III 22 - Chair: Juan Pedro Mellado, Max Planck Meteorological Institute

3:35PM L1.00001 Experimental study of turbulence induced wall temperature fluctuations[1]

, ANIRBAN GARAI, JAN KLEISSL, Dept. of Mechanical and Aerospace Engineering, University of California, San Diego, BOUNDARY LAYER LATE AFTERNOON AND SUNSET TURBULENCE COLLABORATION — Turbulent heat transport is critical in engineering applications and atmospheric flows. The relative strength of background shear and buoyancy near the wall influences coherent structures responsible for much of the heat transport. Previous studies show that shear dominated flow causes streaky-like structures; whereas buoyancy dominated flow causes cell-like structures. In this work, we investigated the influence of flow structures on the wall temperature and heat flux in a convective atmospheric boundary layer. Turbulence data at different heights and high frequency wall temperature were obtained during the Boundary Layer Late Afternoon and Sunset Turbulence field campaign at Lannemezan, France from 7 June – 8 July, 2011. Conditional averaging confirms that the warm wall causes warm ejection events, and cold sweep events cause cooling of the wall. The wall temperature structures move along the wind and their advection speed is close to the wind speed of the upper logarithmic layer and mixed layer, have a size of about 0.2 times the boundary layer depth, become streakier with stability and its standard deviation follows a -1/3 power law with stability parameter, Obukhov length.

[1]We are thankful to all Boundary Layer Late Afternoon and Sunset Turbulence field campaign participants for data sharing and funding from a NASA New Investigator Program award.

3:48PM L1.00002 Surface-Pressure Fluctuations due to Coherent Microscale Processes in the Atmospheric Boundary Layer[1]

, GREGORY LYONS, NATHAN MURRAY, University of Mississippi - NCPA — Recent work has found evidence for microscale coherent turbulence in the atmospheric boundary layer due to the breakdown of Kelvin-Helmholtz billows. It is hypothesized that these structures, which advect with the mean flow, may be identified by transient fluctuations they cause in the boundary wall pressure. Surface-pressure fluctuations beneath the turbulent atmospheric boundary layer were measured at Reese Technology Center, located in the Llano Estacado mesa region of West Texas. Two multi-element arrays of piezoelectric bimorph infrasound sensors, selected for their sensitivity to frequencies as low as 0.1 Hz, were used to measure the pressure fluctuations. The boundary layer was simultaneously profiled up to 200 meters in elevation by an on-site meteorological instrument tower. All measurements were taken continuously for 44 hours. To study coherent propagating events, methods of beamforming and model reduction were applied to the surface-pressure fluctuations. Through dynamic mode decomposition, wavelike eigenfunctions of the pressure dynamics are identified that propagate with either advective or acoustic speeds.

[1]Supported by DOE Award No. DE-EE0003269

4:01PM L1.00003 On the characterization of coherent structures within a neutrally-stratified atmospheric boundary layer

, GIUSEPPE ROSI, BENEN LE BASTIDE, University of Calgary, JULIA GAEBLER, University of Calgary, Technische Universitaet Berlin, MATIAS KINZEL, California Institute of Technology, DAVID RIVAL, University of Calgary — Up to this point, a clear characterization of wind turbulence and extreme gust events through experimentation has frustrated countless researchers. The statistical analysis of fluctuating components has been exhausted while the conditional analysis of extreme events, though insightful, often results in constricted conclusions that cannot be bridged from study to study. Thus the current study shifts towards an understanding of the fundamental turbulent flow structures within a neutrally-stratified atmospheric boundary layer. Two approaches to characterize coherent wind structures are presented. The first approach identifies hairpin-vortex heads by correlating three-dimensional, fluctuating data from two high-speed anemometers situated at 40m and 50m heights on a wind mast. The model assumes that a hairpin-vortex head can be approximated as a transverse vortex with a Vatistas viscous core of assumed radius when the hairpin-vortex head impinges onto the two anemometers. The second approach employs large-scale particle tracking velocimetry to follow seeded bubbles next to the wind mast. The results obtained with both approaches are then compared, and the advantages and shortcomings of each method are discussed.

4:14PM L1.00004 Nested large-eddy simulations of nighttime shear-instability waves and transient warming in a steep valley[1]

, BOWEN ZHOU, FOTINI CHOW, University of California, Berkeley — This numerical study investigates the nighttime flow dynamics in a steep valley. The Owens Valley in California is highly complex, and represents a challenging terrain for large-eddy simulations (LES). To ensure a faithful representation of the nighttime atmospheric boundary layer (ABL), realistic external boundary conditions are provided through grid nesting. The model obtains initial and lateral boundary conditions from reanalysis data, and bottom boundary conditions from a land-surface model. We demonstrate the ability to extend a mesoscale model to LES resolutions through a systematic grid-nesting framework, achieving accurate simulations of the stable ABL over complex terrain. Nighttime cold-air flow was channeled through a gap on the valley sidewall. The resulting katabatic current induced a cross-valley flow. Directional shear against the down-valley flow in the lower layers of the valley led to breaking Kelvin-Helmholtz waves at the interface, which is captured only on the LES grid. Later that night, the flow transitioned from down-slope to down-valley near the western sidewall, leading to a transient warming episode. Simulation results are verified against field observations and reveal good spatial and temporal precision.

[1]Supported by NSF grant ATM-0645784.

4:27PM L1.00005 Delay in convection in nocturnal boundary layer due to aerosol-induced cooling

, DHIRAJ KUMAR SINGH, V.K. PONNULAKSHMI, G. SUBRAMANIAN, K.R. SREENIVAS, EMU (JNCASR) — Heat transfer processes in the nocturnal boundary layer (NBL) influence the surface energy budget, and play an important role in many micro-meteorological processes including the formation of inversion layers, radiation fog, and in the control of air-quality near the ground. Under calm clear-sky conditions, radiation dominates over other transport processes, and as a result, the air layers just above ground cool the fastest after sunset. This leads to an anomalous post-sunset temperature profile characterized by a minimum a few decimeters above ground (Lifted temperature minimum). We have designed a laboratory experimental setup to simulate LTM, involving an enclosed layer of ambient air, and wherein the boundary condition for radiation is decoupled from those for conduction and convection. The results from experiments involving both ambient and filtered air indicate that the high cooling rates observed are due to the presence of aerosols. Calculated Rayleigh number of LTM-type profiles is of the order 10^5-10^7 in the field and of order 10^3-10^5 in the laboratory. In the LTM region, there is convective motion when the Rayleigh number is greater than 10^4 rather than the critical Rayleigh number ($Ra_c = 1709$). The diameter of convection rolls is a function of height of minimum of LTM-type profiles. The results obtained should help in the parameterization of transport process in the nocturnal boundary layer, and highlight the need to accounting the effects of aerosols and ground emissivity in climate models.

4:40PM L1.00006 Direct Numerical Simulation of the Convective Boundary Layer[1]

, JADE RACHELE GARCIA, JUAN PEDRO MELLADO, Max Planck Institute for Meteorology — The inversion of the dry shear-free Convective Boundary Layer is investigated by means of Direct Numerical Simulation (DNS). This work is motivated by the importance of entrainment and related mechanisms at the inversion of the atmospheric boundary layer, combined with the uncertainty of Large-Eddy Simulations (LES) there. Despite moderate Reynolds numbers attainable, results show that the achieved scale separation is enough to capture the expected $1/2$ power law evolution in time, and the expected structure—an inversion-capped outer layer, whose statistics are comparable to LES results and atmospheric data when normalized with the convective scales, and an inner layer near the surface comparable to that of the heated plate case. In agreement with some previous investigations, the entrainment ratio A is a factor of two less than the nominal 0.2 even though the corresponding entrainment velocity is within 5% of the Zero-order Model prediction with $A = 0.2$. To understand this apparent discrepancy, we use DNS data to directly determine the behavior of the terms of an exact equation for the entrainment ratio.

[1]Jülich Research Centre for the computing time

4:53PM L1.00007 DNS of stratified spatially-developing turbulent thermal boundary layers , GUILLERMO ARAYA, LUCIANO CASTILLO, Department of Mechanical Engineering, National Wind Resource Center, Texas Tech University, KENNETH JANSEN, Aerospace Engineering Sciences, University of Colorado — Direct numerical simulations (DNS) of spatially-developing turbulent thermal boundary layers under stratification are performed. It is well known that the transport phenomena of the flow is significantly affected by buoyancy, particularly in urban environments where stable and unstable atmospheric boundary layers are encountered. In the present investigation, the Dynamic Multi-scale approach by Araya et al. (JFM, 670, 2011) for turbulent inflow generation is extended to thermally stratified boundary layers. Furthermore, the proposed Dynamic Multi-scale approach is based on the original rescaling-recycling method by Lund et al (1998). The two major improvements are: (i) the utilization of two different scaling laws in the inner and outer parts of the boundary layer to better absorb external conditions such as inlet Reynolds numbers, streamwise pressure gradients, buoyancy effects, etc., (ii) the implementation of a Dynamic approach to compute scaling parameters from the flow solution without the need of empirical correlations as in Lund et al (1998). Numerical results are shown for ZPG flows at high momentum thickness Reynolds numbers ($\sim$ 3,000) and a comparison with experimental data is also carried out.

5:06PM L1.00008 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 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. We report hotwire measurements in a plane normal to the flow direction at various downstream positions and free stream velocities to examine the development and formation of the artificial ABL. 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 by Hultmark in 2010 for high Reynolds number flat plate turbulent boundary layers.

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

5:19PM L1.00009 Global Intermittency in Stably Stratified Turbulent Ekman Layers[1] , STIMIT SHAH, Princeton University, CHARLES MENEVEAU, Johns Hopkins University, ELIE BOU-ZEID, Princeton University — Current weather models rely on similarity theories to represent fluxes in the lower atmosphere that assume turbulence is relatively homogeneous in space and time. In addition, many SGS models for LES assume turbulence is homogeneous to apply planar-averaging when computing model coefficients.However, under very stable conditions turbulence can be intermittent, resulting in inhomogeneity in space. Measurements in the atmosphere under very stable conditions have also shown oscillatory behavior of turbulent quantities (Coulter 90;Van de Weil et al 01). A number of possible mechanisms causing this have been described previously (Hunt 85;Nappo 91;Mahrt 99). In this study we investigate the dynamics of intermittency and oscillatory behavior using DNS, in order to improve our understanding and turbulence closures. For very stable cases, TKE production is found to drop down to zero resulting in laminarization of the flow field. This causes a drop in wall stress, which in turn allows the flow to accelerate, thus increasing shear culminating with increased production of TKE. The period of variations in TKE from high to low and back is on the order of a few hours, as found in the atmosphere. Dependencies of the period and other properties of the variations on Richardson number are explored.

[1]Stimit Shah and Elie Bou-Zeid are funded by NSF under AGS-1026636 and Charles Meneveau is supported by NSF-AGS-1045189

5:32PM L1.00010 The critical layer for gravity waves in sheared rotating stratified flows , CHRISTOPHE MILLET, CEA, DAM, DIF, F-91297 Arpajon, France, FRANCOIS LOTT, LMD-CNRS, ENS Paris, F-75231 Paris cedex 5, France — We re-examined the propagation of gravity waves through a critical layer surrounded by two inertial levels in the case of a constant vertically sheared flow. This problem involves a transition from balanced (where the quasi-geostrophic approximation applies) to sheared gravity waves. The three-dimensional disturbance is described analytically using both an exact solution and a WKB approximation valid for large Richardson numbers. In contradiction with past studies which show that there is finite reflection and did not analyse the transmission (Yamanaka and Tanaka, 1984), we find that reflection is extremely too small to be significant. The reasons that previous authors made incorrect evaluations are related to the fact that (i) the equations yielding to these results are extremely involved and (ii) the values of reflection and transmission coefficients are exponentially small or null, e.g. quite difficult to cross check numerically. Interestingly, these values are exactly like in the much simpler non-rotating case analysed by Booker and Bretherton (1966). Some practical implications for the problem of the emission of gravity waves by potential vorticity anomalies, analysed recently in Lott et al. (2013), are also discussed.

Monday, November 19, 2012 3:35PM - 5:45PM –

Session L2 Convection and Buoyancy-Driven Flows VI: Plumes 23A - Chair: Andrew Wells, University of Oxford

3:35PM L2.00001 Collapsing plumes and resurrecting fountains , TON VAN DEN BREMER, Department of Engineering Science, University of Oxford, GARY HUNT, Department of Civil and Environmental Engineering, Imperial College London — We explore the range of behaviour predicted for steady plumes and fountains that undergo an increase or decrease in buoyancy which arise due to phase changes or chemical reactions. We model these changes in the simplest possible way by assuming a quadratic relationship between the density and the temperature of the fluid. We thereby extend the model of Caulfield & Woods ('95) to include the most recent developments in the literature on steady releases of buoyancy emitted vertically from horizontal area sources in unconfined quiescent environments of uniform density based on the plume model of Morton, Taylor & Turner ('56). We provide closed-form solutions and identify four classes of solution: collapsing plumes, resurrecting fountains, plumes with enhanced buoyancy and fountains with enhanced negative buoyancy. We provide criteria for each category of behaviour in terms of the source-value of two non-dimensional quantities: the Richardson number and a temperature parameter.

Caulfield, C. C. P. & Woods, A. W. 1995. Plumes with non-monotonic mixing behaviour. Geophys. Astro. Fluid. **79**, 1-4, 173-199. Morton, B.R., Taylor, G.I. & Turner, J.S. 1956. Turbulent gravitational convection from maintained and instantaneous sources. Proc. Roy. Soc. Lond. A **234**, 1-23.

3:48PM L2.00002 ABSTRACT WITHDRAWN –

4:01PM L2.00003 A Comparison of Single and Multiphase Turbulent Jets, Pure and Forced Plumes at Moderate Reynolds Numbers.[1] , G.N. TAUB, S. BALACHANDAR, University of Florida, F. PLOURDE, Ecole National Superieure De Mecanique et D'Aerotechnique, France — Turbulent axisymmetric shear flows, such as jets and plumes arise often in industrial applications and environmental studies. The recent Deep Water Horizon oil spill in the Gulf of Mexico is one example which brought to light the need for a greater understanding of the turbulent behavior of such flows. The results of Direct Numerical Simulations of single phase pure jets (Re=2000), pure plumes ($Gr = 2000^2$) and forced plumes (Re=1684, Ri=0.025), where both buoyancy and initial momentum are present, will be compared and contrasted. In addition to the mean flow behavior, second and third order statistics will be presented as well as the turbulent energy balance for all three flows. In the case of the forced plume, the transition from jet like behavior near the source of initial momentum to plume like behavior in the far field will be discussed. Preliminary results of laboratory experiments and Direct Numerical Simulations of multiphase forced plumes will also be presented.

[1]Supported in part by the National Science Foundation (NSF OISE-0968313) through the Partnership for International Research and Education (PIRE).

4:14PM L2.00004 Instability of plumes driven by localized heating , FRANCISCO MARQUES, Univ. Politecnica de Catalunya - UPC, JUAN M. LOPEZ, School of Mathematical and Statistical Sciences, Arizona State Univ, YOUNGHAE DO, Department of Mathematics, Kyungpook National Univ. — Plumes due to localized buoyancy sources are of wide interest due to their prevalence in many situations, including fires, chimneys, volcanoes, deep sea hydrothermal vents and a wide variety of other atmospheric and oceanic situations. In this study, we are interested in the transition from laminar to turbulent plumes. In experiments, this transition is found to be sensitive to external perturbations. Therefore a well-controlled set-up has been chosen: a localized heat source at the bottom of an enclosed cylindrical container, at a uniform temperature except for the heat source. At moderate Rayleigh numbers Ra, the flow consists of a plume, which is steady, axisymmetric and purely poloidal. By increasing Ra, the flow undergoes a supercritical Hopf bifurcation at Ra 3.8E7, to an axisymmetric "puffing" plume, that becomes unstable to three-dimensional disturbances at about Ra 5.4E7. At larger Ra $>$ 1.E8, the plume becomes chaotic via a torus-breakup bifurcation.

4:27PM L2.00005 Laminar plume formation by high pressure CO2 , FRANCOIS NADAL, CEA CESTA, Le Barp, France, PATRICE MEUNIER, IRPHE, Aix Marseille Univ., CNRS, Marseille, France, BERNARD POULIGNY, ERIC LAURICHESSE, CRPP, CNRS, Univ. Bordeaux, France — Convection flows have often revealed the presence of plumes, especially in the earth's mantle where the Schmidt number is large. There has thus been a large number of studies on plumes created by a point source. However, there are very few results on plumes generated by an extended source. Here, we present experimental, numerical and theoretical results on the flow created by high pressure CO2 dissolved into distilled water. The thin layer of dense fluid created at the surface destabilizes through the Rayleigh-Taylor instability and leads to a laminar and parallel stationary plume. The plume width and amplitude are measured by Particle Image Velocimetry for various aspect ratios, Bond and Rayleigh numbers. They are in good agreement with the numerical result if a no-slip boundary condition is assumed at the free surface. Finally, the theory for a plume generated by a point source is adapted for an extended source, which leads to different scaling exponents (with a logarithmic dependence), in excellent agreement with the experimental and numerical results. This study thus provides a simple and accurate description of axisymmetric plumes generated by an extended source.

4:40PM L2.00006 Self-similarity of Boussinesq Miscible Thermals: an Experimental Study , BING ZHAO, NTU, ADRIAN LAI, SMART, ADRIAN LAW, NTU, ERIC ADAMS, MIT — The gross characteristics of fully-developed round miscible thermals have been well studied and reported to be self-similar (e.g. Scorer, 1957). However, there have been very few studies (Bond & Johari, 2005; Hart, 2008) concerning the internal structures of the thermal. Many important questions related to the interior fluid dynamics inside the thermal, including the self-similarity of the internal velocity and scalar distributions, remain outstanding. In the present study, detailed PIV and PLIF measurements were conducted in the axisymmetric plane (i.e. side view) of a negatively buoyant Boussinesq thermal to reveal the detailed internal structures, with CCD cameras that synchronized with a unique release mechanism that minimized the initial variations. Synchronized simultaneous flow visualization (with spotlights and a video camera) were also made to monitor the developmental shape of the thermal through a bottom view. The simultaneous information enabled an objective assessment of the experimental quality. The results showed that the maximum radius of the miscible thermal grows linearly with travel distance, which agrees with previous studies using dimensional analysis with self-similarity. The radius of the vortex ring is found to be expanding linearly, but surprising at a smaller growth rate that the overall thermal size. This raises a critical question whether the self-similarity with thermals truly exists or not. The results will be presented at the meeting.

4:53PM L2.00007 The velocity field of laboratory fire whirls , KATHERINE HARTL, STACY GUO, ALEXANDER SMITS, Princeton University — Fire whirls increase the intensity of wild fires, and they enhance the spreading rate of fire by ejecting flaming debris, often rendering prepared fire fighting plans useless. Fire whirls are produced by buoyancy, ambient vorticity, and the titling of regions of intense horizontal negative shear at the fire front. They can shed from a smoke plume, form in the turbulent wake of a hill, or arise due to concentrated heat sources. To understand and model fire whirls in greater detail, an experimental study is conducted to generate a fire whirl in the laboratory, and examine the velocity fields outside and inside the vortex core. We use flow visualization and PIV to obtain qualitative and quantitative data, and discuss the scaling behavior.

5:06PM L2.00008 Laboratory-Scale Simulation of Spiral Plumes in the Mantle. , ALBERT SHARIFULIN, Perm State Technical University, ANATOLY POLUDNITSIN, Perm State University — On the basis of laboratory simulation a mechanism is established for the formation of the upper mantle convection spiral plumes from acore hot point in the presence of a roll-type large-scale convective flow. The observed plume has horizontal sections near the upper limit of cavity, which may lead to the formation of chains of volcanic islands. We experimentally simulated the appearance of a plume from the hot by green laser generated hot spot and study its interaction with cellular flow, simulating beneath the plates shear flow. It is shown that the presence of cellular convective motion may lead to the formation of a strange spiral convective plume. Experimentaly 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 [1].

[1] Skilbeck, J. N.; Whitehead, J. A. (1978). Formation of discrete islands in linear chains. Nature **272**: 499–501. http://en.wikipedia.org/wiki/Nature_(journal)

5:19PM L2.00009 Meltwater-plume dynamics under an evolving ice shelf , ANDREW WELLS, University of Oxford — Recent observations and models suggest that melting at the base of floating ice shelves can significantly impact ice sheet flow, with consequences for sea level rise. As a simplified analogue of ice-shelf basal melting, I consider a theoretical model of the melting of a two-dimensional stationary ice-shelf above a warmer glacial ocean. Melting rates are controlled by a turbulent buoyancy-driven plume of meltwater that is coupled to an ice-water free boundary that evolves as a result of melting. The evolving slope of the ice-shelf base provides a feedback on flow and heat transfer in the meltwater plume, with potential consequences for the stability of ice shelves and other systems featuring a coupling of melting and buoyancy-driven flow.

5:32PM L2.00010 Generation of gravity waves by convective plumes , MICHAEL LE BARS, IRPHE, CNRS and Aix-Marseille University, France, and Department of Earth and Space Sciences, UCLA, USA, STEPHANE PERRARD, PETER HUCK, ADRIEN AUBERT, PATRICE LE GAL, IRPHE, CNRS and Aix-Marseille University, France — In many geo- and astrophysical situations, a turbulent convective fluid layer is separated from a stably stratified one by a relatively sharp but deformable interface. Examples include the oceanic upper mixed layer and underlying pycnocline zone, the convective and radiative zones in stars, the atmospheric convective layer and overlying stratosphere. Here we present 2 experimental studies to get a global description of the properties of the excited wave field and its interaction with the convective motions. In the first set-up, a turbulent plume generated by injection of water impinges upon the interface between a uniform density layer and a linearly stratified one of salted water. In the second set-up, we take benefit from the unusual property of water that its density has a maximum value near 4°C to study its convective and oscillatory motions in a tank with a bottom boundary at about 0°C and an hotter upper surface. In both experiments, the velocity fields are measured non-intrusively using PIV, allowing to determine the energy flux extracted from the plume and transported by the waves. The spectral analysis of the signals exhibit preferred values of the frequency and wavelength of excited waves which depend on the location. Those results compare well with a simple analytical approach where the impinging plume is modeled as a initial displacement of the interface with a Gaussian shape in time and space.

Monday, November 19, 2012 3:35PM - 5:32PM –

Session L3 Multiphase Flows: Atomization and Droplets 23B - Chair: Olivier Desjardins, Cornell University

3:35PM L3.00001 Droplet Jumping Induced by Focused Surface Acoustic Wave on Superhydrophobic Surface[1] , MARTEN DARMAWAN, Sungkyunkwan University, KWANGSUN JEON, Enjet, Inc., DOYOUNG BYUN, Sungkyunkwan University — We investigate the droplet jumping phenomena that are induced by focused surface acoustic wave on superhydrophobic surface. The utilization of an identical pair of single phase unidirectional transducers (SPUDTs) leads to the focusing of acoustic wave energy on a small region between them. This focused energy gives a very high acceleration as well as rapid interface destabilization to the liquid droplet and thus derives the jumping phenomenon once surpasses some Weber number's threshold value. We intriguingly investigate the effect of the small contact area of droplet on superhydrophobic surface, which is generated by using plasma treatment, to the droplet jetting phenomena and how it deforms under this circumstance. Furthermore, a parametric study, i.e. varying acoustic energy power, volume of droplet and degree of arc SPUDT, is also performed to investigate their effect on the elongated jumping droplet.

[1]Partially funded by Basic Science Research Program through the National Research Foundation of Korea(NRF, 2011-0016461) and the Industrial Core Technology Development Project through the Ministry of Knowledge and Commerce.

3:48PM L3.00002 Convective Air Mass Transfer in Submerged Superhydrophobic Surfaces , CHRISTINA A. BARTH, MOHAMED A. SAMAHA[1], HOOMAN VAHEDI TAFRESHI, MOHAMED GAD-EL-HAK, Virgina Commonwealth University — Under pressure and flowing water, air entrapped in submerged superhydrophobic coatings eventually dissolve into the water. We analyze from first principles a simple mass transfer problem. We introduce an *effective* slip to a Blasius boundary layer, and solve the hydrodynamic as well as the mass transfer equations. Similarity solutions are found for both systems of equations. We then solve the two-dimensional problem of alternating free-shear and no-slip regions. This situation simulates spanwise microridges. The second problem has no similarity solution but is solvable using approximate integral methods. A mass-transfer correlation is achieved and relates the surface geometry (or gas area fraction) to the effective slip. The analytical results are compared to numerical simulations obtained via the FLUENT software for the laminar Navier–Stokes equations. Longevity, or time-dependent hydrophobicity, could be estimated for the simple laminar, flat-plate geometry investigated, and is compared to experimental results.

[1]Presently at Princeton University

4:01PM L3.00003 ABSTRACT WITHDRAWN –

4:14PM L3.00004 Efficient High-Fidelity Simulation of Pressure Swirl Injection , MARK OWKES, OLIVIER DESJARDINS, Cornell University — Atomization of hydrocarbon fuels is of critical importance to air-breathing, liquid-fueled internal combustion engines used in the transportation sector. While atomizers can take widely different forms and operate under a wide range of conditions, pressure swirl atomizers are essential as they represent the main component of most air-blast spraying systems for gas turbines. Experimental investigations of pressure swirl injectors are challenging due to their often-intricate geometry, and due to the formation of a conical liquid sheet that complicates optical access to the central injector region. Hence, numerical simulations have the potential to shed light on the physics of such injectors. To tackle such a task computationally, methods are needed for efficiently handling of complex geometries, and robustly and conservatively handle high-density ratio turbulent two-phase flows. In this work, a computational study of pressure swirl injection is presented based on an efficient approach that combines ideas from immersed boundary, level set, volume of fluid, and ghost fluid methods. The flow dynamics within the injector as well as the spray cone are discussed.

4:27PM L3.00005 High fidelity simulation of transcritical injection , MARIOS SOTERIOU, HUI GAO, XIAOYI LI, DUSTIN DAVIS, United Technologies Research Center — Transcritical injection of a multi-component fluid occurs in many practical applications such as diesel and rocket engines. In this type of injection a liquid fuel at a supercritical pressure but subcritical temperature, is introduced into an environment where conditions are supercritical. The convoluted physics of the transition from the subcritical to the supercritical state is linked to thermodynamic property variations and poses challenges to numerical simulation. For example, the temporary presence of surface tension implies that both the subcritical liquid-vapor interface and the transition boundary to supercritical fluid need to be captured. In this work, numerical simulation of a binary system of a subcritical liquid injecting into a supercritical, quiescent gaseous environment is performed. A coupled level set and volume of fluid method is adopted to capture the liquid-vapor interface, across which the continuity of mass and energy fluxes is preserved. The fluid state over the range of subcritical liquid to supercritical fluid is determined by incorporating the Peng-Robinson equation of state. To efficiently account for the sharp changes in properties near the liquid-vapor interface and the transition boundary to supercritical fluid, an adaptive mesh refinement technique is employed. Analysis of results focuses on the impact of vanishing surface tension as conditions transition from sub-critical to supercritical.

4:40PM L3.00006 ABSTRACT WITHDRAWN –

4:53PM L3.00007 Isogeometric analysis of drop deformation in isoviscous shear flow , AMIN AHMADI JONEIDI, CLEMENS VERHOOSEL, PATRICK ANDERSON, Eindhoven university of technology — We use the Boundary Integral Method (BIM) to study the deformation of a drop in iso-viscous shear flow. Traditionally the drop surface is represented by a linear triangular mesh. The novelty of this work compared to prior studies is applying Isogeometric Analysis (IGA) to define the drop interface. In this method splines are used as smooth shape functions to create the surface instead of the traditional non-smooth triangular surface. This makes IGA applicable in the case when the physics at the interface becomes more complicated, for example if the deformation of a red blood cell or a vesicle is investigated; these involve higher-order surface gradients in the force jump across the interface. For the iso-viscous drop it is observed that the drop deforms and deviates from the initial spherical shape and orients itself in the fixed direction. Different values of the capillary number -which is the measure of the ratio between viscous and surface tension forces- have been studied and the results match very well with traditional BIM. IGA results for more complex interfacial force jumps are discussed.

5:06PM L3.00008 Impact of Liquid Fuel Boundary Condition and Nozzle Geometry on Liquid Jet in Crossflow Atomization[1] , SINA GHODS, MARCUS HERRMANN, Arizona State University — The atomization of a liquid jet by a high speed cross-flowing gas has many applications such as gas turbines and augmentors. The mechanisms by which the liquid jet initially breaks up, however, are not well understood. Experimental studies suggest the dependence of spray properties on operating conditions and nozzle geometry. Detailed numerical simulations can offer better understanding of the underlying physical mechanisms that lead to the breakup of the injected liquid jet. In this work, we present detailed numerical simulation results of turbulent liquid jets injected into turbulent gaseous cross flows for different liquid fuel boundary conditions and injector geometries. We employ a finite volume, balanced force fractional step flow solver to solve the Navier-Stokes equations coupled to a Refined Level Set Grid method to follow the phase interface. To enable the simulation of atomization of high density ratio fluids, we ensure discrete consistency between the solution of the conservative momentum equation and the level set based continuity equation by employing the Rescaled Conservative Momentum method. We analyze the impact of liquid jet turbulent fluctuations and injector geometry on different jet properties such as jet penetration and generated drop sizes.

[1]This work was supported by NSF grant number CBET-0853627

5:19PM L3.00009 Measurement of liquid film flow on nuclear rod bundle in micro-scale by using very high speed camera system[1] , SON PHAM, ZENSAKU KAWARA, TAKEHIKO YOKOMINE, TOMOAKI KUNUGI[2], Department of Nuclear Engineering, Graduate School of Engineering, Kyoto University — Playing important roles in the mass and heat transfer as well as the safety of boiling water reactor, the liquid film flow on nuclear fuel rods has been studied by different measurement techniques such as ultrasonic transmission, conductivity probe, etc. Obtained experimental data of this annular two-phase flow, however, are still not enough to construct the physical model for critical heat flux analysis especially at the micro-scale. Remain problems are mainly caused by complicated geometry of fuel rod bundles, high velocity and very unstable interface behavior of liquid and gas flow. To get over these difficulties, a new approach using a very high speed digital camera system has been introduced in this work. The test section simulating a 3x3 rectangular rod bundle was made of acrylic to allow a full optical observation of the camera. Image data were taken through Cassegrain optical system to maintain the spatiotemporal resolution up to 7 μm and 20 μs. The results included not only the real-time visual information of flow patterns, but also the quantitative data such as liquid film thickness, the droplets' size and speed distributions, and the tilt angle of wavy surfaces. These databases could contribute to the development of a new model for the annular two-phase flow.

[1]Partly supported by the Global Center of Excellence (G-COE) program (J-051) of MEXT, Japan.
[2]Corresponding Author

Monday, November 19, 2012 3:35PM - 5:45PM –

Session L4 Drops VIII 23C - Chair: Francisco Higuera, E.T.S. Ingenieros Aeronauticos, UPM

3:35PM L4.00001 Measurement of interfacial tension of ionic liquid-dielectric liquid system using the shape of an electrically deformed droplet , DONG WOOG LEE, DO JIN IM, IN SEOK KANG, POSTECH — When a charged droplet of an ionic liquid is translated by uniform electric field, it is deformed into a prolate shape. The deformed shape is used for estimation of the interfacial tension of ionic liquid-dielectric liquid system. The typical Debye length of an ionic liquid is of sub-nanometer scale, while the typical radius of a droplet is about 0.5 mm. Therefore, from the electrostatics view point, the droplet can be approximated as a perfect conductor. In addition, the net charge accumulated at the droplet surface does not cause any significant change of the composition of ionic liquid. Thus, no change of interfacial tension is expected to be due to the electrical effect. The experimental configuration of the present study has also other advantages. The first is that the Reynolds number is smaller than unity and the Stokes flow limit can be applied. In this small Reynolds number regime, uniform streaming flow does not cause shape change up to the first order shape deformation. This means that to this order, the shape change is solely due to the electrical effect. The second is that net charge of a conducting droplet does not cause shape change up to the first order deformation. This means that we do not need the information about the amount of net charge of a droplet. In the experiments, the ionic liquids 1-alkyl-3-methylimidazolium alkyl sulfate ([Cn-mim] [CH3SO4] and [Cn-min] [C2H5SO4], n=2, 4, 6) are used as the droplet phase and the silicone oil KF-54(500cS) is used as the continuous phase.

3:48PM L4.00002 Charge Transfer into Aqueous Droplets via Kilovolt Potentials , B.S. HAMLIN, E.R. ROSENBERG, W.D. RISTENPART, Dept. Chemical Engineering and Materials Science, Univ. California Davis — When an aqueous droplet immersed in an insulating oil contacts an electrified surface, the droplet acquires net charge. For sufficiently large field strengths, the charged droplet is driven back and forth electrophoretically between the electrodes, in essence "bouncing" between them. Although it is clear that the droplet acquires charge, the underlying mechanism controlling the charge transfer process has been unclear. Here we demonstrate that the chemical species present in the droplet strongly affect the charge transfer process into the drop. Using two independent charge measurement techniques, high speed video velocimetry and direct current measurement, we show that the charge acquired during contact is strongly influenced by the droplet pH. We also provide physical evidence that the electrodes undergo electroplating or corrosion for droplets with appropriate chemical species present. Together, the observations strongly suggest that electrochemical reactions govern the charge transfer process into the droplet.

4:01PM L4.00003 Numerical and asymptotic analysis of an electrified jet[1] , SANTIAGO E. IBÁÑEZ, F.J. HIGUERA, ETS Ingenieros Aeronauticos, UPM, Madrid, Spain — A numerical study is presented of a liquid jet of finite electrical conductivity issuing at a constant flow rate from a metallic capillary set at a high voltage relative to a distant electrode. The electric field due to the applied voltage induces a conduction current in the liquid that accumulates electric charge at its surface and gives rise to electric stresses that stretch the jet. A quasi-unidirectional model of the flow is proposed and boundary elements and finite difference methods are used to compute the electric field, the flow of the liquid and the distribution of surface charge as functions of the applied voltage, the flow rate and the physical properties of the liquid. Asymptotic results for large and small values of the flow rate are worked out. At small values of the flow rate, the electric current increases as the square root of the flow rate and the surface of the liquid resembles a cone followed by a thin jet. The electric field at the surface attains a maximum in a certain current transfer region where convection of the surface charge becomes the dominant contribution to the electric current. At large values of the flow rate, the electric current and the length of the current transfer region become independent of the flow rate.

[1]The authors acknoledge support from the Spanish Plan Nacional de I+D through project and scholarship DPI2010-20450-C03-01.

4:14PM L4.00004 Periodic emission of droplets from an electrified meniscus , A.J. HIJANO, I.G. LOSCERTALES, Universidad de Malaga, Spain, S.E. IBANEZ, F.J. HIGUERA, E. T. S. Ingenieros Aeronauticos, UPM, Madrid, Spain — We report an experimental characterization of the periodic emission of mass from a meniscus hanging from the tip of a needle, fed at a constant liquid flow rate, when subjected to a constant high voltage with respect to a grounded plate. For a given flow rate, the potential is swept until a periodic regime is reached. Two very distinct emission modes are observed to coexist. In each pulsation, the meniscus elongates under the electric field until it develops a conical cusp from which a highly charged fine aerosol is continuously emitted at a very high frequency, resembling the cone-jet electrospray. Simultaneously, the meniscus develops a neck which eventually forms a droplet, much larger than those emitted from the unsteady electrospray. The detachment of the large droplet occurs at frequencies of the order of the inverse of the liquid capillary time, whereas the fine droplets are emitted at much higher, unmeasurable frequencies. For the low viscosity, highly conducting liquids used in this study, most of the mass emitted per pulse is carried by the large droplets, whereas the charge is transported by the fine aerosol. Scaling laws for the dimensionless pulsation frequency and large droplet diameter are provided in terms of an electric Bond number and a dimensionless flow rate.

4:27PM L4.00005 Electro-deformation of a surfactant-laden viscous drop[1] , HERVE NGANGUIA, YUAN-NAN YOUNG, New Jersey Institute of Technology, PETIA VLAHOVSKA, Brown University, JIA ZHANG, HAO LIN, Rutgers University — In this work we investigate the equilibrium shape and dynamics of a surfactant-laden viscous drop under an electric field. The full Taylor-Melcher leaky dielectric framework is employed. We use both small-deformation theory (for a drop slightly deformed from a spherical shape) and semi-decomposition method for a highly deformed (prolate or oblate) spheroidal drop. Both theoretical approaches are validated by comparing predicted deformation with experimental data by Ha and Yang (1998). The dependence of the critical Capillary number for the equilibrium shape on the surfactant coverage is quantified, and more detailed analysis of the models shed light on the surfactant effects on the drop deformation under an electric field.

[1]NSF

4:40PM L4.00006 Whipping in electrified liquid jets , JOSEFA GUERRERO MILLAN, VENKAT GUNDABALA, ALBERTO FERNANDEZ-NIEVES, Georgia Institute of Technology — Whipping is a non-axisymmetric instability that appears in electrified jets. In air, it usually manifests in a chaotic fashion and thus, its structure and properties have been hard quantify experimentally. We use electro-coflow to generate a steady-state whipping structure and quantify its geometry and how it depends on operating parameters, like liquid flow rates and applied voltage.

4:53PM L4.00007 Magnetic actuation of immersed coupled droplets: Experiments and simulations , JOSEPH OLLES, AMIR HIRSA, Rensselaer Polytechnic Institute, KRISHNARAJ SAMBATH, OSMAN BASARAN, Purdue University — A system of two droplets connected through a cylindrical hole in a plate, with pinned contact lines, shows promise in several engineering applications including fast adaptive optics, microscale actuators and pumps, and adhesion devices. Such coupled droplets, surrounded by a passive gas medium, have been studied extensively. With the motivation of advancing this technology, here we consider coupled-droplet systems comprised of ferrofluid immersed in an immiscible liquid. Nonlinear characteristics of the system are studied by exciting the ferrofluid using a small electromagnet at various frequencies. The responses, tracked by observing the interface motion through high-speed imaging, are analyzed. Fluid velocities measured using index matched PIV techniques are also characterized. To corroborate the experimental results, the ferrofluid system with a magnetic force on the coupled droplets is simulated with the magnetic force approximated by a uniform body force. The axisymmetric Navier-Stokes system, which governs the flow in both the ferrofluid and surrounding fluid, is solved using the Galerkin finite element method. By aligning the simulations with experimental data, a novel method of extracting interfacial fluid properties is elucidated. Operational parameters where experiments are contradicted by simulations are also discussed.

5:06PM L4.00008 Electrohydrodynamics of suspension of liquid drops in AC fields , MD. ABDUL HALIM, ASGHAR ESMAEELI, Southern Illinois University Carbondale — Manipulation of liquid drops by an externally applied electric field is currently the focus of increased attention because of its relevance in a broad range of industrial processes. The effect of a uniform DC electric field on a solitary drop is well studied; however, less is know about the impact of electric field on suspension of liquid drops, and very little information is available on the impact of AC field on a single or a suspension of drops. Here we report the results of Direct Numerical Simulations of electrohydrodynamics of suspension of liquid drops. The governing equations are solved using a front tracking/finite difference technique, in conjunction with Taylor's leaky dielectric model. The imposed electric potential comprises of two parts, a time-independent base and a time-dependent part. The goal is to explore the relative importance of these two components in setting the statistically steady state behavior of the suspension. To this end, we report the results of three sets of simulations, where (i) the time-dependent part act as a perturbation on the base potential, (ii) the two components are of the same order, and (iii) the time-dependent part is much larger than the base potential. The problem is studied as a function of the governing nondimensional parameters.

5:19PM L4.00009 Electrohydrodynamic suspension of liquid drops in microgravity , ALI BEHJATIAN, ASGHAR ESMAEELI, Southern Illinois University Carbondale — Electrohydrodynamics of drops are currently the focus of increased attention. The majority of these studies, however, ignore the effect of gravity to better isolate the electric field effects. Here we are interested in exploring the effect of this parameter on the dynamics. To this end, we perform Direct Numerical Simulations to study dynamics of suspension of leaky dielectric drops in DC electric fields. The governing equations are solved using a front tracking/finite difference scheme in conjunction with Taylor's leaky dielectric theory. The electric-field induced flow and drop deformation is studied as a function of the governing nondimensional numbers of the problem. The buoyancy is considered as a perturbation parameter and the behavior of the individual drops and suspensions at the three different regions of the deformation-circulation is explored.

5:32PM L4.00010 Influencing the In-flight Shape and Velocity of a Ferrofluid Drop by a Magnetic Field: Case of a Falling Drop Towards a Surface , ALIDAD AMIRFAZLI, J.N. WU, University of Alberta, MIGUEL CABRERIZO-VILCHEZ, University of Granada — In this work magnetic field generated either by a solenoid or a permanent magnet was used to manipulate and change the shape of a drop approaching a surface. The magnetic field was also used to change the velocity of the drop approaching the surface. The capability to sculpt the drop shape and change its velocity opens up new ways of manipulating drop impact onto a surface, which can be interesting for printing industry, especially 3D printing for manufacturing parts. EFH1 (Ferrotec, USA), a colloidal dispersion of magnetite in an oil, was used as the ferrofluid, and its drops were generated with a size of $\sim$2.4 mm using a dispensing system. High speed imaging and image processing were the primary tools for this study allowing data acquisition, and analysis, respectively. Results showed that the in-flight drop shape can be changed from spherical (no field applied) to mildly elliptical or even cylindrical depending on the method of magnetic field generation, the strength of the magnetic field, and the duration of application of the magnetic field (when solenoid was used). Drop velocities could also be increased by up to three times of what would have been possible under free fall condition for a drop. Finally a discussion of in-flight drop breakup as an ultimate way to change the drop shape and its potential for applications will be provided.

Monday, November 19, 2012 3:35PM - 5:45PM –

Session L5 Computational Fluid Dynamics VI 24A - Chair: Robert Moser, University of Texas at Austin

3:35PM L5.00001 Periodic Cavitation in a High-Speed Water Inducer at an Off-Design Flow Coefficient , RYAN LUNDGREEN, RYAN CLUFF, DANIEL MAYNES, STEVEN GORRELL, Brigham Young University, KERRY OLIPHANT, Concepts-sNREC — Time resolved numerical simulations were conducted on a high-speed water inducer designed to operate under cavitating conditions at both on and off-design flow rates. A segregated solver was employed and the turbulence model was the realizable k-epsilon approach. The solution discretization is second order accurate in space and first order accurate in time. Cavitation within the domain becomes periodic as the cavitation number decreases. At flow coefficients smaller than the design flow coefficient, a large time-varying volume of cavitation is observed upstream of the inducer causing the system to become unstable for practical use. Large regions of reversed flow at the blade tip cause the incoming fluid to increase in velocity and the effective mass flow area to decrease. It is this increase in velocity that leads to the formation of the periodic vapor cavity upstream of the inducer. The vapor cavity increases in size until it completely blocks the core of the passage, forcing the flow out toward the shroud. As the flow near the shroud accelerates, the reversed flow at the blade tip decreases and the vapor cavity decreases in size until it collapse completely, causing a large jump in pressure throughout the entire flow domain.

3:48PM L5.00002 Comparison: on-design and off-design flow through a high speed turbo pump inducer , RYAN CLUFF, RYAN LUNDGREEN, STEVE GORRELL, DANIEL MAYNES, Brigham Young University, KERRY OLIPHANT, Concepts NREC — A computational fluid dynamic comparison was performed between on-design and off-design flow rates through a four-bladed axial turbopump inducer. Using CD Adapco's Star-CCM as the CFD package an analysis of the two flow-rate cases was made. The simulations were run time-resolved and with two phases (water and water vapor). Turbulence employed the realizable k-epsilon model and cavitation was predicted using the Rayleigh-Plasset model. The solution discretization is second order accurate in space and first order accurate in time. The results show classical breakdown curves for both flow-rate cases. Breakdown is the condition where the entire flow path in the inducer becomes filled with vapor and the head rise over the inducer is decreased dramatically. Both cases experience breakdown at about the same cavitation number; however, because the off-design case generally has a larger head rise, its breakdown occurs at higher back pressures than the on-design case. Additionally, the off-design case experiences larger amounts of incidence that result in regions of reversed flow along the shroud and an increase of instabilities throughout the machine. Performance maps will be discussed comparing the two cases on head rise and efficiency.

4:01PM L5.00003 Optimal design of solenoid valve to minimize cavitation by numerical analysis , SEUNGBIN KO, ILHOON JANG, SIMON SONG, Hanyang University — Keeping pace with the development of clean energy, hybrid cars and electric vehicles are getting extensive attention recently. In an electronic-control brake system which is essential to those vehicles, a solenoid valve is used to control external hydraulic pressure that boosts up the driver's braking force. However, strong cavitation occurs at the narrow passage between the ball and seat of a solenoid valve due to sudden decrease in pressure, leading to severe damage to the valve. In this study, we investigate the cavitation numerically to discover geometric parameters to affect the cavitation, and an optimal design to minimize the cavitation using optimization technique. As a result, we found four parameters: seat inner radius, seat angle, seat length, and ball radius. Among them, the seat inner radius affects the cavitation most. Also, we found that preventing a sudden reduction in a flow passage is important to reduce cavitation. Finally using an evolutionary algorithm for optimization we minimized cavitation. The optimal design resulted in the maximum vapor volume of fraction of 0.04 while it was 0.7 for reference geometry.

4:14PM L5.00004 Transient High-Pressure Fuel Injection Processes , DORRIN JARRAHBASHI, WILLIAM A. SIRIGNANO, University of California — The transient behavior of the jet emerging from the orifice during the start-up and shut-down portion of the injection is addressed. Use has been made of an unsteady axisymmetric code with a finite-volume solver of the Navier-Stokes equations for liquid streams and adjacent gas and a level-set method for liquid/gas interface tracking. The acceleration of the liquid during start-up is about 10^6 m/s^2 at the orifice exit. When the jet emerges from the orifice, drag forces due to the dense ambient air cause a deceleration. Also, the dynamic protrusions from the jet surface created by Kelvin-Helmholtz instability are subject to local accelerations that lead to Rayleigh-Taylor instability. The higher the Weber and the Reynolds numbers, the shorter the unstable surface wavelengths which appear; so, the more challenging is the resolution problem. Where resolving the entire jet becomes computationally expensive, we examine stream-wise segments of the jet, treating these segments as ballistic slugs coming from the orifice. This reduction of the computational domain is designed to give the required resolution to characterize the physics through our computations.

4:27PM L5.00005 Effect of a Magnetic Field on Turbulent Flow in Continuous Casting Mold[1] , RAMNIK SINGH, PRATAP VANKA, BRIAN G. THOMAS, University of Illinois — Electromagnetic Braking (EMBr) fields are applied to control the turbulent mold flow for defect reduction in continuous steel casting. The effect of EMBr depends on the path of induced electric current which is modified by presence of the highly conducting solidifying shell. The mold geometry is complex involving flow in a high-aspect ratio closed channel with bifurcated jet impinging obliquely on the side walls. The extremely transient nature and the anisotropic behavior of turbulence under the EMBr field make numerical studies challenging. We use large eddy simulations to study effects of EMBr with electrically insulating and conducting boundary conditions. Magnetohydrodynamic equations are solved using a fractional step method with second order spatial and temporal accuracy. The electric potential method is used as magnetic Reynolds number is low for liquid metal flows. The solver was first validated with measurements from scaled GaInSn model and simulations were then performed to study real casters at industrial conditions. Time averaged and transient behaviors of the flow were studied by collecting distributions of mean velocities, turbulent fluctuations and vorticity. The simulations reveal that the electrical boundary conditions have a major effect on the flow structure.

[1]National Science Foundation Grant CMMI 11-30882

4:40PM L5.00006 Adjoint Airfoil Optimization of Darrieus-Type Vertical Axis Wind Turbine , ROMAN FUCHS, HENRIK NORDBORG, HSR University of Applied Sciences, Rapperswil — We present the feasibility of using an adjoint solver to optimize the torque of a Darrieus-type vertical axis wind turbine (VAWT). We start with a 2D cross section of a symmetrical airfoil and restrict us to low solidity ratios to minimize blade vortex interactions. The adjoint solver of the ANSYS FLUENT software package computes the sensitivities of airfoil surface forces based on a steady flow field. Hence, we find the torque of a full revolution using a weighted average of the sensitivities at different wind speeds and angles of attack. The weights are computed analytically, and the range of angles of attack is given by the tip speed ratio. Then the airfoil geometry is evolved, and the proposed methodology is evaluated by transient simulations.

4:53PM L5.00007 Simulation of a valveless pump with an elastic tube , SOO JAI SHIN, KAERI, CHEONG BONG CHANG, HYUNG JIN SUNG, KAIST — A valveless pump consisting of a pumping chamber with an elastic tube was simulated using an immersed boundary method. The interaction between the motion of the elastic tube and the pumping chamber generated a net flow toward the outlet throughout a full cycle of the pump. The net flow rate of the valveless pump was examined by varying the stretching coefficient, bending coefficient, the aspect ratio of the elastic tube, and the frequency of the pumping chamber. As the stretching and bending coefficients of the elastic tube increased, the net flow through the valveless pump decreased. Elastic tubes with aspect ratios in the range of $2<l/d<3$ generated a higher flow rate than that generated for tubes with aspect rations of $l/d=1$ or 4. As the frequency of the pumping chamber increased, the net flow rate of the pump for $l/d=2$ increased. However, the net flow rate for $l/d=3$ was nonlinearly related to the pumping frequency due to the complexity of the wave motions. Snapshots of the fluid velocity vectors and the wave motions of the elastic tube were examined over one cycle of the pump. The relationship between the average gap in the elastic tube and the average flow rate of the pump was analyzed.

5:06PM L5.00008 Numerical Study of Thermoacoustic Spontaneous Oscillations in an Axisymmetric Closed Tube , KATSUYA ISHII, ITC, Nagoya University, SHUN KITAGAWA, KOICHIRO SHIRAI, CSE, Nagoya University, SHIZUKO ADACHI, Tokyo International University — We study the stability of thermoacoustic spontaneous oscillations (Taconis oscillations) by numerically solving the axisymmetric compressible Navier-Stokes equations. The flow fields in a cylindrical closed tube are simulated by the block pentadiagonal scheme with the second–order time marching and the fourth-order convective term. We consider the helium gas in the closed tube with both hot end parts and a cold central part. When the temperature ratio is larger than the critical value, the spontaneous oscillation is observed. In the case of r(the length ratio of the hot part to the cold part)=1, we observe the fundamental mode of a standing wave as the spontaneous oscillation. On the other hand, in the case of $r < 0.42$ three different oscillation modes are observed: the fundamental mode and the second mode of a standing wave, and the oscillation with a shock wave.

5:19PM L5.00009 Numerical Analysis of Transport Phenomena for the Design of the Ejector in a PEM Fuel Cell , ELHAM HOSSEINZADEH, MASOUD JABBARI, MASOUD ROKNI, Department of Mechanical Engineering, Technical University of Denmark, Nils Koppels Allé, 2800 Kgs. Lyngby, Denmark — In the present study, Computational Fluid Dynamics (CFD) technique is used to design an ejector for anode recirculation in a specific automotive PEMFC system. A CFD model is firstly established and tested against well-documented and relevant solutions from literature, and then used to different ejector geometries under different working conditions. Results showed that one ejector with the optimized geometry cannot cover the required recirculation in the entire range of the current, and having two ejector for different range of currents is a new proposed alternative, in which the system can take a better advantage of ejector for recirculation purpose.

5:32PM L5.00010 Numerical Study of 3D Flow and Mixing Properties in the Rotated Arc Mixer[1] , ESUBALEW ALEMAYEHU DEMISSIE, MICHEL SPEETJENS, HERMAN CLERCX, Eindhoven University of Technology, GUY METCALFE, Commonwealth Scientific and Industrial Research Organisation (CSIRO) — Laminar mixing of fluids is an important process in many industrial operations. However, insight into 3D flow and mixing in the devices remains limited. This is largely due to their complex construction, which makes experimental investigation difficult and representation by analytical solutions impossible. This motivates the current numerical study on essentially 3D flow and mixing properties in a device representative of a wide class of mixers: the Rotated Arc Mixer (RAM). Key aspects to be investigated are transient effects between consecutive mixing cells and role of fluid inertia. Two RAMs, comprising of 5 and 10 cells, have been investigated by resolving full 3D Navier-Stokes equations. Simulations exposed small backflow zones near to entrance (exit) of each cell. They also revealed that up to 40% of cell length is involved in flow transition. However, the extent of this transition depends on flow parameters and cell geometry. The flow retains global spatial periodicity of the mixer. Moreover, essentially 3D internal symmetries within each cell exist in Stokes limit. Poincaré sections show that the transient and inertia effects cause a change in the location and size of non-mixing zones. This implies a significant impact of these effects on the mixing properties.

[1]The authors greatly appreciate the support by Dutch Technology Foundation (STW).

Monday, November 19, 2012 3:35PM - 5:45PM –

Session L6 Microfluidics: Capillary I 24B - Chair: Pirouz Kavehpour, University of California, Los Angeles

3:35PM L6.00001 Breakup of liquid filaments on a partially wetting solid substrate , GIOVANNI GHIGLIOTTI, Department of Mathematics, University of British Columbia, Vancouver, B.C., Canada V6T 1Z2, CHUNFENG ZHOU, Corning Incorporated, One Science Center Drive, Corning, NY 14831, JAMES J. FENG, Department of Chemical and Biological Engineering, University of British Columbia, Vancouver, B.C., Canada V6T 1Z3 — We report direct numerical simulations of liquid filaments breaking up into droplets on partially wetting substrates. It is motivated by recent experiments, linear stability analyses, and lubrication-based calculations. The fluid flow is governed by the Stokes equations and the contact line motion is handled by a phase-field model, which also serves to capture the interfacial motion. The coupled Stokes and Cahn-Hilliard equations are solved using a finite-element algorithm in three dimensions. This avoids additional approximations of the fluid flow or contact line motion, and allows us to consider arbitrary contact angles $0 \leq \theta \leq \pi$ on the substrate. We simulate both the breakup of infinite liquid filaments via growing capillary waves and that of finite liquid filaments with drops pinching off from the ends. The results show the differences in the two processes and in the final drop size, spacing and polydispersity, as well as the dependence of these two phenomena on the value of the contact angle with the substrate. The development of capillary waves agrees well with prior linear analysis and the end-pinching results offer new insights into this poorly understood phenomenon.

3:48PM L6.00002 ABSTRACT WITHDRAWN –

4:01PM L6.00003 Investigation of drag reduction properties of liquid impregnated microtextured surfaces , VAMSI SPANDAN, Indian Institute of Technology Madras, GARETH MCKINLEY, Massachusetts Institute of Technology, SARIT DAS, Indian Institute of Technology Madras — The liquid repelling and drag reducing properties of superhydrophobic surfaces are attributed to microscopic pockets of air trapped in between topographical structures with low surface energy. In this work we numerically investigate the drag reduction properties of textured surfaces which are impregnated with an immiscible liquid lubricant instead of air. Although these surfaces outperform conventional superhydrophobic surfaces in dynamic characteristics such as droplet movement (due to lack of contact line pinning), the question whether such surfaces can exhibit drag reduction when immersed in a fluid remains open. We employ a continuum approach (using Volume of Fluid simulations) to investigate changes in skin friction on such surfaces. This method enables the simulation of two incompressible, immiscible fluids with the inclusion of capillary effects between the phases and specification of a finite contact angle between the phases and the textured wall. We study the effect of viscosity ratio, interfacial tension, and topography of the microtextures on the drag reducing properties of such liquid impregnated surfaces.

4:14PM L6.00004 Wetting transitions on silicon nanowires with different surface functionalizations , XIUMEI XU, GUY VEREECKE, ERIK VAN DEN HOOGEN, JENS SMEERS, SILVIA ARMINI, TINNE DELANDE, HERBERT STRUYF, IMEC — The wetting property of an ideal smooth surface is well described by Young's equation, on the other hand, the wetting behavior on a patterned substrate can be quite different due to the topographic influence. To date the transition from Wenzel or hemi-wicking state to the metastable Cassie-Baxter wetting state is still an active research topic. In literature the patterned substrates have a typical feature size in the micrometer scale, and it has been found that although a complete wetting is achievable on such dimensions, there can be a Cassie state of wetting in mesoscopic scale due to the air trapping by the nanometric roughness on the features. In this work we investigate the wetting properties of nano-patterned substrates. Silicon nanopillars (30-40 nm in diameter and 90 nm in pitch) are fabricated with different heights ranging from 70 to 450 nm, and different treatments are applied to functionalize the pillar surface to have a good coverage of water contact angles. Three wetting regimes are observed: hemi-wicking state for contact angles lower than 50 degrees, Cassie-Baxter state for contact angles larger than 80 degrees, and in between there is a sharp transition with the apparent contact angles increasing from 30 to 150 degrees. Wenzel model does not agree with the measurements for the entire range of the contact angle, indicating the wetting mechanism might be very different in the nanometer scale.

4:27PM L6.00005 Wetting and spreading behaviors of impinging microdroplets on textured surfaces[1] , DAE HEE KWON, SANG JOON LEE, Center for Biofluid and Biomimic Research, Department of Mechanical Engineering, POSTECH, CENTER FOR BIOFLUID AND BIOMIMIC RESEACH TEAM — Textured surfaces having an array of microscale pillars have been receiving large attention because of their potential uses for robust superhydrophobic and superoleophobic surfaces. In many practical applications, the textured surfaces usually accompany impinging small-scale droplets. To better understand the impinging phenomena on the textured surfaces, the wetting and spreading behaviors of water microdroplets are investigated experimentally. Microdroplets with diameter less than 50 μm are ejected from a piezoelectric printhead with varying Weber number. The final wetting state of an impinging droplet can be estimated by comparing the wetting pressures of the droplet and the capillary pressure of the textured surface. The wetting behaviors obtained experimentally are well agreed with the estimated results. In addition, the transition from bouncing to non-bouncing behaviors in the partially penetrated wetting state is observed. This transition implies the possibility of withdrawal of the penetrated liquid from the inter-pillar space. The maximum spreading factors (ratio of the maximum spreading diameter to the initial diameter) of the impinging droplets have close correlation with the texture area fraction of the surfaces.

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

4:40PM L6.00006 Precipitation of salt in saline water drop on superhydrophobic surface , BONGSU SHIN, Seoul National University, MYOUNG-WOON MOON, Korea Institute of Science and Technology, HO-YOUNG KIM, Seoul National University — In the membrane distillation process, water vapor of heated, pressurized saline water is transported across the membrane to be collected as pure water. While the water-repellency of the membrane surface has been considered an important parameter affecting the distillation efficiency, the resistance of the membrane to the contamination due to salt has gathered little scientific interest thus far. Here we experimentally investigate the precipitation of salt in sessile saline water drops, to find drastic differences in salt crystallization behavior depending on the water-repellency of solid surface. On a moderately hydrophobic surface with a static contact angle with water being about 150 degrees, salt crystals are aligned and stacked along the initial contact line, forming an interesting structure resembling an igloo. On a superhydrophobic surface with about 164 degrees of static contact angle with water, salt crystallizes only at the center of the drop-solid contact area, forming a pebble-shaped structure. We explain this difference by comparing the evaporation modes (constant contact radius versus constant contact angle) of the sessile drops on those surfaces. We also visualize the liquid flow within drops undergoing evaporation and precipitation at the same time using PIV.

4:53PM L6.00007 Pressure and Heating Effects on Superhydrophobic Friction Reduction , TAE JIN KIM, SUNGYUN HANN, CARLOS HIDROVO, The University of Texas at Austin — Slip in internal flows is known to reduce friction and thus decrease the required pumping power. One method to achieve slip is by roughening the surface to induce Cassie state (a phenomenon in which a liquid rests on top of a rough surface with a gas layer formed underneath). While most work in this area has concentrated on optimizing the surface microtexturing geometry to maximize the friction reduction effects, less attention has been paid to the effects of partially wetted conditions. Our research goal is to control/track the air-water interface location within the roughness elements and study the interface effects on the microchannel friction. The frictional behavior of the flow suggests that (1) the air-water interface resembles closer to a no-slip boundary than a shear-free one, (2) the friction is rather insensitive to the degree of microtexturing wetting, and (3) the fully wetted microtexturing provides lower friction than the non-wetted ones. In accordance to the high frictional nature of the air-water interface, the effective slip length was varied by controlling the location of the air-water interface through heating. Results have shown that the Cassie state can be maintained at higher pressures through heating, but the flow may be pinched if excessive heat is applied to the microchannel.

5:06PM L6.00008 Facile fabrication of super-hydrophobic surfaces with 3D pillar structures , SHENGJIE ZHAI, HUI ZHAO, YINGTAO JIANG, University of Nevada Las Vegas — Super-hydrophobic surfaces have attracted growing interest due to their unique properties, including drag reduction, facilitation of heat transfer, self-cleaning, anti-corrosion, anti-sticking, and anti-contamination. However, the method of fabricating super-hydrophobic surfaces with regular 3D micro/nano pillars structures is still complicated. Here we present a simple, reliable, and low-cost fabrication method which can create complex 3D structures. Briefly, the commercial nanostamping products like CD, DVD,and bluray disc serve as the PDMS mold The pit size (LxWxH) of CD, DVD, and Blueray is 0.8μm x 0.15μm x 0.1μm, 0.4μm x 0.15μm x 0.1μm, and 0.15μm x 0.15μm x 0.1μm. The PDMS surface with the relevant structures can be directly replicated from the molds by the soft lithography technology. The precise geometric structures including height, width, and density of pillar arrays can be readily controlled by using different optical discs. The contact angle is measured about 136-140 degree. We also study the relationship between the contact angle and different feature size. Finally, we measure the slip length for different structures

5:19PM L6.00009 Friction Reduction and Robustness for Laminar Fluid Flow on Spray-Coated Superhydrophobic Mesh Surfaces , SIDDARTH SRINIVASAN, Massachusetts Institute of Technology, WONJAE CHOI, The University of Texas at Dallas, KYOO-CHUL PARK, SHREERANG CHHATRE, ROBERT COHEN, GARETH MCKINLEY, Massachusetts Institute of Technology — We measure the effective Navier slip length for flow over a liquid-repellent non-wetting surface (fabricated using a spray-deposition technique) which supports a composite solid-liquid-air interface. The morphology of the hydrophobic textured substrate consists of randomly distributed corpuscular microstructures that stabilize a layer of trapped air upon immersion in liquid. The reduction in viscous skin-friction due to this "plastron layer" is evaluated using torque measurements in a parallel plate rheometer, and results in measured slip lengths of b_{slip} $\approx$40 μm, that are comparable to the mean periodicity of the microstructure. The use of dual-textured spray-coated woven meshes increases the magnitude of the effective slip length to between b_{slip} $\approx$ 90 μm to 200 μm depending on the mesh dimensions. We compute the wetted-solid fraction φ_s from surface evolver simulations, and we demonstrate that the experimentally obtained slip-lengths are consistent with the logarithmic prediction of Davis & Lauga. Finally we define a robustness parameter (A^*) to quantify the stability of the plastron. And illustrate the inverse correlation between A^* and b_{slip} by means of a design chart.

5:32PM L6.00010 Spreading of Emulsions on Glass Substrates , ALIREZA MOHAMMAD KARIM, PIROUZ KAVEHPOUR, University of California, Los Angeles — The wettability of emulsions is an important factor with explicit influence in an extensive variety of industrial applications ranging from the petroleum to food industries. Surprisingly, there is no comprehensive study of emulsion spreading to date; this is due to the complexity of the structure of the emulsions and non-homogeneity of the dispersed phase bubbles in size as well as distribution through the emulsion. The spreading of water/silicone oil emulsions on glass substrates was investigated. The emulsions were prepared with varying volume fractions of water dispersed in silicone oil, with addition of small amounts of surfactant to stabilize the emulsion structure. The time dependent variation of dynamic contact angle, base diameter, and the spreading rate of the droplets of an emulsion are different from a pure substance. The effect of water/silicone oil weight percentage as well as the droplet size and dispersed phase bubble size were also investigated. The weight percentage of water/silicone oil emulsion and droplet size did not have significant influence on the spreading dynamics; however the dispersed phase drop size affected the spreading dynamics substantially.

Monday, November 19, 2012 3:35PM - 5:45PM –

Session L7 Non-Newtonian Flows II 24C - Chair: Alexander Morozov, University of Edinburgh

3:35PM L7.00001 Dynamics of non Newtonian vortex rings , C.A. PALACIOS-MORALES, C. BARBOSA, R. ZENIT, Universidad Nacional Autonoma de Mexico — The dynamics of formation and evolution of non-Newtonian vortex rings generated in a piston-cylinder arrangement are studied. The ratio of the piston displacement L_m to the internal cylinder diameter D_0, the piston velocity U_p and fluid properties determine the vortex properties and evolution. Measurements of the 2D velocity field were obtained with a PIV technique. The vortex circulation Γ was computed considering a vortex identification scheme (Q criterion). Experiments with fluids with different rheological properties (shear thinning and viscoelastic) are presented. Our Newtonian experiments agree with previous investigations. For shear-thinning liquids, we observed that the final vortex circulation decreases with the fluid power index, n. We show that the total circulation ejected from the cylinder is reduced when the thinning property of the liquid increases; thus, the circulation confined inside the vortex ring, is reduced too. For vortex rings in a viscoelastic liquid, the formation of a 'negative wake' (returning flow) and a second vortex ring with opposite whirl are observed. We show that the negative wake results from the high extension rates produced during the vortex formation.

3:48PM L7.00002 Flow Behavior near the Liquid-to-Solid Transition , H. HENNING WINTER, University of Massachusetts Amherst — Amorphous materials are studied experimentally in search for viscosity/elasticity properties in the approach of the liquid-to-solid transition from the liquid side (LSTLS). Two vastly different processes are considered, gelation and glass transition. While viscoelasticity in incipient gels is dominated by the shorter relaxation modes, expressed in a decaying relaxation time spectrum, it is the opposite in a glass forming liquid near LSTLS which exhibits an advancing. For both classes of amorphous materials, the relaxation time spectrum broadens significantly near LSTLS and it shares the same powerlaw format. Still, the relaxation behavior differs fundamentally for the two material classes since the powerlaw exponent is positive for the glass transition and negative for gelation, i.e. the relaxation patterns of gelling fluids and glass formers are inverse near LSTLS. The entire study is founded in Boltzmann's constitutive equation of linear viscoelasticity; the stress is caused by a wide range of modes where, as argued here, short modes dominate gelation and long modes dominate the glass transition. Several examples are shown for each class of materials in order to test the proposed transition behavior for glasses (colloidal and molecular) on the one side and chemical/physical gels on the other. One of the results of this experimental study is that it provides a decisive criterion that distinguishes the glass transition from gelation.

4:01PM L7.00003 Bending and buckling of viscoplastic threads , IAN HEWITT, NEIL BALMFORTH, University of British Columbia — We use a slender body theory to describe the dynamics of a thin viscoplastic thread undergoing extrusion, such as occurs when squeezing toothpaste from a tube. The theory adopts the Bingham model for a yield stress fluid, together with an asymptotic approximation for the stress and strain-rate profiles across the narrow width of the thread, which imply that the thread must either be rigid or fully yielded across its entire width. A compact description of the resultant longitudinal stress and moment acting on the thread allows these yielded and unyielded regions to be identified for given external forces. The theory is applied to extrusion flows; the yield stress prevents any deformation until a critical length of extrusion is reached, after which the dynamically evolving yielded regions mediate a distinctive drooping of a horizontal beam, or a catastrophic collapse of an upright beam.

4:14PM L7.00004 Dynamics of Viscoplastic Sheets , NEIL BALMFORTH, University of British Columbia — A theory is presented for the dynamics of slender sheets of viscoplastic fluid, equivalent to classical analysis of elastic beams and viscous plates. The effect of a yield stress is highlighted. The theory is applied to the fall of a liquid bridge supported at its two ends. The yield stress halts the fall of the bridge; the final asymptotic state is calculated in various limits.

4:27PM L7.00005 Transient Non-Newtonian Screw Flow , NARIMAN KHORASANI, Payame Noor University — The influence of axial flow on the transient response of the pseudoplastic rotating flow is carried out. The fluid is assumed to follow the Carreau-Bird model and mixed boundary conditions are imposed. The four-dimensional low-order dynamical system, resulted from Galerkin projection of the conservation of mass and momentum equations, includes additional nonlinear terms in the velocity components originated from the shear-dependent viscosity. In absence of axial flow the base flow loses its radial flow stability to the vortex structure at a lower critical Taylor number, as the pseudoplasticity increases. The emergence of the vortices corresponds to the onset of a supercritical bifurcation which is also seen in the flow of a linear fluid. However, unlike the Newtonian case, pseudoplastic Taylor vortices lose their stability as the Taylor number reaches a second critical number corresponding to the onset of a Hopf bifurcation. Existence of an axial flow, manifested by a pressure gradient appears to further advance each critical point on the bifurcation diagram. In addition to the simulation of spiral flow, the proposed formulation allows the axial flow to be independent of the main rotating flow. Complete transient flow field together with viscosity maps are also presented.

4:40PM L7.00006 Numerical Simulation of Nanoparticle Clustering with Experimental Validation[1] , ZHIGANG FENG, GREGORY SLOAN, KIRAN BHAGANAGAR, University of Texas at San Antonio, DEBYJOTI BANERJEE, Texas A&M University — In this study a numerical approach for modeling the forces acting on nanoparticles was performed and the dynamics of transient nanoparticle agglomeration have been explored. The validity of the approach is demonstrated by examining a pair of nanoparticles in a fluid. The force interactions due to the presence of the electric double layer (EDL) were identified as a significant factor in determining the propensity for agglomerative of the nanoparticles. Simulations were performed to demonstrate the clustering and agglomeration of an ensemble of nanoparticles. The simulation results provide an estimate for the time scale for the agglomeration and the resultant structure of the agglomerated ensemble of nanoparticles. Subsequently simulations were performed using this numerical model corresponding to the available experimental data in the literature. The predictions from the numerical simulations show that the change in zeta potential (determined in part by the pH of the solvent phase) is a crucial parameter that affects the level of agglomeration of the nanoparticles.

[1]Supported by the Natioanl Science Foundation under award number HRD-1137764.

4:53PM L7.00007 Numerical modeling of flowing soft materials , FEDERICO TOSCHI, Eindhoven University of Technology, The Netherlands, ROBERTO BENZI, University of Rome Tor Vergata, Italy, MASSIMO BERNASCHI, IAC-CNR, Rome, Italy, PRASAD PERLEKAR, Eindhoven University of Technology, The Netherlands, MAURO SBRAGAGLIA, University of Rome Tor Vergata, Italy, SAURO SUCCI, IAC-CNR, Rome, Italy — The structural properties of soft-flowing and non-ergodic materials, such as emulsions, foams and gels shares similarities with the three basic states of matter (solid, liquid and gas). The macroscopic properties are characterized by non-standard features such as non-Newtonian rheology, long-time relaxation, caging effects, enhanced viscosity, structural arrest, hysteresis, dynamic disorder, aging and related phenomena. Large scale non-homogeneities can develop, even under simple shear conditions, by means of the formation of macroscopic bands of widely different viscosities ("shear banding" phenomena). We employ a numerical model based on the Lattice Boltzmann method to perform numerical simulations of soft-matter under flowing conditions. Results of 3d simulations are presented and compared to previous 2d investigations.

5:06PM L7.00008 Brownian Dynamics Simulations of Flow Induced Conformation of Single Polymer Chains , ALPARSLAN OZTEKIN, EDWARD WEBB, FRANK ZHANG, XUANHONG CHENG, Lehigh University — Coarse-grained molecular dynamics simulation of single polymer chains is conducted to examine flow induced conformational changes of single polymer chains in shear and extensional flows. Dynamic properties of polymeric molecules are described using bead-spring models; these models put coarse grain groups of atoms into single particles, connected to adjacent particles via spring like interactions. A common representation of the bonded interaction, or spring, is given by the Finitely Extensible Non-linear Elastic model. The constants used in the model are determined form single-molecule force spectroscopic experiments. Simulations and experiments will help to achieve a clear picture of how von Willebrand Factor, a blood clotting protein, model system for flow-induced activation, achieves flow sensing at the single protein/polymer level. The model includes bead-bead interactions, hydrodynamic interaction, the volume of the beads and spring-spring interaction for finitely extensible dumbbell and other spring models. The effects of walls on the dynamics of polymer chains are also presented. The results are presented for various chain lengths and flow conditions. Hydrodynamic interaction and the presence of wall have strong influences on the dynamics of the polymer chain.

5:19PM L7.00009 Computational modeling of dilute biomass slurries , MICHAEL SPRAGUE, JONATHAN STICKEL, National Renewable Energy Laboratory, PAUL FISCHER, Argonne National Laboratory, JAMES LISCHESKE, National Renewable Energy Laboratory — The biochemical conversion of lignocellulosic biomass to liquid transportation fuels involves a multitude of physical and chemical transformations that occur in several distinct processing steps (e.g., pretreatment, enzymatic hydrolysis, and fermentation). In this work we focus on development of a computational fluid dynamics model of a dilute biomass slurry, which is a highly viscous particle-laden fluid that can exhibit yield-stress behavior. Here, we model the biomass slurry as a generalized Newtonian fluid that accommodates biomass transport due to settling and biomass-concentration-dependent viscosity. Within a typical mixing vessel, viscosity can vary over several orders of magnitude. We solve the model with the Nek5000 spectral-finite-element solver in a simple vane mixer, and validate against experimental results. This work is directed towards our goal of a fully coupled computational model of fluid dynamics and reaction kinetics for the enzymatic hydrolysis of lignocellulosic biomass.

5:32PM L7.00010 Stability of streamwise vortices in shear flows of viscoelastic fluids , ALEXANDER MOROZOV, School of Physics & Astronomy, University of Edinburgh — Recent work on transient growth in viscoelastic shear flows [1] suggests that even though these flows are linearly stable, a non-modal perturbation could be sufficiently amplified resulting in long-living transient disturbances. Similar to Newtonian shear flows, these disturbances take form of the streamwise vortices. Here we perform a linear stability analysis of the streamwise vortices superimposed on the steady shear flow. We find that this flow is linearly unstable towards 3D perturbations. In Newtonian case, this instability is known to play an important role in sustaining exact coherent structures [2] that dynamically organise the transition to turbulence. We discuss the possibility that similar structures exist in viscoelastic shear flows.

[1] M. Jovanovic and S. Kumar, Phys. Fluids 22, 023101 (2010)
[2] F. Waleffe, Phys. Fluids 9, 883 (1997)

Monday, November 19, 2012 3:35PM - 5:45PM –

Session L8 Drops IX 25A - Chair: Ying Sun, Drexel University

3:35PM L8.00001 Bag Breakup of Viscous Drops , VARUN KULKARNI, School of Mechanical Engineering, Purdue University, DANIEL GUILDENBECHER, Sandia National Laboratories, STEPHANIE FIREHAMMER, School of Aeronautics and Astronautics, Purdue University, PAUL SOJKA, School of Mechanical Engineering, Purdue University — Secondary breakup of drops has been of interest since the early 1900s. The present work focuses on the drop fragmentation process in the presence of a continuous gas - jet at *We* corresponding to the bag breakup regime. Its purpose is to extend current understanding of inviscid drops to the viscous case through a combination of theoretical and experimental efforts. Various aspects of the physical process, such as regime boundaries, drop fragment sizes, and initiation time, which have been hitherto mostly empirical, are studied. The theoretical formulations are based on conservation equations and hydrodynamic linear stability analysis. Techniques which involve extensive testing using PDA and high speed imaging are employed to compare model predictions with experimental data. The breakup event, as visualized through the bag expansion extent is seen to occur at a slower rate than its inviscid counterpart and is captured adequately by theory. This revealed the reasons for *Oh* dependence. Also seen is the emergence of a bimodal droplet size distribution corresponding to the rim and bag fragments. Finally, the extent of bimodality was seen to be dependent on *Oh*.

3:48PM L8.00002 Self-similar breakup of near-inviscid liquids[1] , ALFONSO A. CASTREJON-PITA, J. RAFAEL CASTREJON-PITA, JOHN R. LISTER, E. JOHN HINCH, IAN M. HUTCHINGS, University of Cambridge — Experimental results are presented for the final stages of drop pinch-off and ligament breakup for different initial conditions. Water and ethanol were used as working fluids. High-speed imaging and image analysis were utilized in order to determine the contraction rate of the thinning neck and the shape of the liquid thread just before the breakup. Our results show that the geometry of the breakup of near-inviscid fluids is self-similar in the domain of simple dripping. We also demonstrate that, independently of the initial conditions, the necking of these liquids scales with $\tau^{2/3}$, asymptotically giving a unique breakup angle of $18.0 \pm 0.4^{\circ}$. Both observations are in complete agreement with previous theoretical predictions. The angle converges towards self similarity like $\tau^{1/2}$, also as predicted.

[1]Project supported by the EPSRC-UK (EP/G029458/1 and EP/H018913/1) and Cambridge-KACST.

4:01PM L8.00003 The Breakup of Water Cylinders Behind Normal Shocks , J.C. MENG, T. COLONIUS, California Institute of Technology — We simulate the drift and breakup of a water cylinder in the flow behind a normal shock. The unsteady Euler equations, closed using the stiffened-gas equation of state, are solved with a compressible, multicomponent, shock- and interface-capturing algorithm. The effects of surface tension and viscosity are negligible at early times compared to the larger shear forces. Computed drift velocities are in good agreement with experiments. For the high- speed flow regimes considered, the breakup mode is stripping. Pressure gradients arise on the cylinder's surface causing it to deform laterally. As the cylinder is flattened, sheets of liquid are drawn off the periphery and break up further downstream. Unsteady vortex shedding is observed in the wake of the disintegrating cylinder. As the shock Mach number is increased, higher airflow velocities result in faster breakup and greater cylinder accelerations. These accelerations are subject to fluctuations that grow with shock strength. Qualitative features of the flow are compared to images from experiments on cylinders and drops.

4:14PM L8.00004 Impact force of a falling drop , DAN SOTO[1], CRISTOPHE CLANET, DAVID QUERE, PMMH / Ladhyx, XAVIER BOUTILLON COLLABORATION[2] — Controlling droplet deposition is crucial in many industrial processes such as spraying pesticides on crops, inkjet printing or spray coating. Therefore, the dynamics of drop impacts have been extensively studied for more than one century. However, few literature describe the impacting force of a drop on a solid flat surface, although it might be a way to measure the size distribution of a collection of falling drops. We investigated experimentally how the instantaneous force at impact depends on impact velocity and drop radius. We also propose a new model to understand our observations.

[1]Physique et Mecanique des Milieux Heterogenes, CNRS, ESPCI, Paris France & Ladhyx, CNRS, Ecole Polytechnique, Palaiseau, France
[2]Laboratoire de Mecanique des Solides, Ecole polytechnique

4:27PM L8.00005 Fabrication of nano-emulsions by bursting bubble at a liquid-liquid interface , JIE FENG, MATTHIEU ROCHÉ, DANIELE VIGOLO, Department of Mechanical and Aerospace Engineering, Princeton University, Princeton NJ 08544, USA, LUBEN ARNAUDOV, SIMEON STOYANOV, Unilever Research & Development, 3133AT Vlaardingen, The Netherlands, HOWARD A. STONE, Department of Mechanical and Aerospace Engineering, Princeton University, Princeton NJ 08544, USA — Bubbles bursting at interfaces is a familiar everyday occurrence and plays a role in important processes of transport across interfaces. Here we demonstrate that the bursting of air bubbles at an air-oil-water interface in the presence of a surfactant and a co-surfactant leads to the dispersion of nano-droplets in water. Using high-speed imaging we investigate the mechanism for the dispersion of objects and show that small droplets detach from the boundary of the bubble towards the bulk water during collapse of the bubble. We also characterize the size and stability of the dispersed objects with dynamic light scattering and microscopy techniques. The observations indicate that a well-defined population of few-hundred-nm-sized droplets is produced by bubble bursting, along with a broad range of sizes above 1 μm. We propose that the dispersed objects are formed because of the rapid motion of the bubble interface during collapse. By varying experimental conditions, we show that the size of the droplets is influenced mainly by the amount of surfactant in the oil phase.

4:40PM L8.00006 Drop impact on a hydrophobic elastic beam , SEAN GART, KATIE NORRIS, DANIEL CHIQUE, SUNGHWAN JUNG, Department of Engineering Science and Mechanics, Virginia Tech — Plant surfaces found in nature often exhibit hydrophobic wetting properties; a particular example is the surface of leaves. Most leaves are compliant enough to survive while being impacted by rain droplets. Here, we investigate this leaf-drop system exhibiting a unique system of coupled elasticity and drop dynamics. By replacing the leaf with a thin piezoelectric cantilever beam, we further measure and harvest this drop kinetic energy as a workable model for an energy-harvester from rain drops.

4:53PM L8.00007 Impact, Rebound, and Deflection of High-Velocity Continuous Droplet Streams , PAUL CHIAROT, JOHN DONOVAN, WILLIAM DOAK, State University of New York at Binghamton — Continuous ink jet (CIJ) systems generate streams of droplets $\sim$30 μm in diameter at rates of up to 350 kHz and velocities in excess of 20 m/s. Diverse applications can benefit from this technology; however, reliable manipulation of the jet, including droplet steering and basic on/off control, remains difficult to achieve. We report a novel strategy to manipulate the trajectory of high-velocity CIJ droplet streams using the dielectrophoretic (DEP) force. Droplet rebound at shallow angles is a key feature of this strategy. Therefore, high-velocity droplet impact with hydrophobic and superhydrophobic surfaces was investigated. Rebound from the hydrophobic surface was governed by the Weber number and impact angle. For the superhydrophobic surface, two distinct operating regimes describe the response of the reflected droplet stream. In the first regime, the droplets remain discrete and uniform after the impact, but exhibit significant deformation and oscillation. In the second regime, the droplets coalesce into a puddle at the surface. The droplet spacing of the incoming stream determines which regime rules; with the critical value a function of the Weber number. In the first regime, an accounting of the kinetic and potential energies reveals that neither droplet oscillation nor rotation can fully account for the loss of translational kinetic energy. This suggests that significant internal circulation must occur in the droplets at impact. A simple dynamic model predicts the trajectory of the droplet streams modified by the DEP force. This work is in collaboration with Dr. Thomas Jones at the University of Rochester.

5:06PM L8.00008 Oscillation and recoil of single and consecutively printed droplets , XIN YANG, VIRAL CHHASATIA, YING SUN, Drexel University — Drops are often used as building blocks for line and pattern printing where their interactions play an important role in determining the morphology and properties of deposited functional materials. In this study, the impact, spreading and oscillation of single and consecutively printed drops on substrates of different wettabilities are examined using a high speed camera. The results show that, for a single droplet impacting at a low Weber number, both the inertia and surface tension play important roles in the initial spreading stage before the droplet starts to oscillate. On a substrate of higher wettability, drop oscillation is damped down faster due to stronger viscous dissipation resulted from a longer liquid oscillation path. It is also found that when a drop impacting on an evaporating sessile drop sitting on a hydrophobic substrate, recoil of the combined drop is observed, in contrast to no recoil for the impact of a single drop under the same condition. Furthermore, a single-degree-of-freedom vibration model for the height of oscillating single and combined drops on a hydrophobic substrate is established. The results show that as viscosity of liquid increases, damping of drop oscillation becomes faster, and the combined drop oscillates longer compared to a single drop.

5:19PM L8.00009 Breakup of impulsively actuated jets from thin films , C. FREDERIK BRASZ, MATTHEW BROWN, Princeton University, YIANNIS VENTIKOS, University of Oxford, CRAIG ARNOLD, Princeton University — Understanding the breakup of liquid jets into droplets is important for printing applications, and much progress has been made on slender jets emanating from nozzles (as in inkjet printing). A more recent alternative to inkjet printing is blister-actuated laser-induced forward transfer (BA-LIFT), in which a laser pulse is absorbed in a polymer layer adjacent to a thin film of ink, forming an expanding blister that ejects ink as a jet. The lack of a nozzle allows for a wider range of inks, and the formation of the jet from an expanding blister in a thin film significantly changes the fluid physics. We study the breakup of these jets computationally by forcing an ink film with a boundary that deforms according to experimental time-resolved measurements of expanding blisters. Computational results are compared with experimental images, and parametric studies explore the effects of varying properties like laser energy, ink viscosity, surface tension, and film thickness on jet breakup. Scaling arguments are presented to justify the observed power laws for these parametric studies.

5:32PM L8.00010 Coalescence, evaporation and particle deposition of consecutively printed colloidal drops , VIRAL CHHASATIA, XIN YANG, JAYMEEN SHAH, YING SUN, Dexel University — In applications such as inkjet printing and spray deposition, colloid drops are often used as building blocks for line and pattern printing where their interactions play important roles in determining the deposition morphology and properties. In this study, the particle deposition dynamics of two consecutively printed evaporating colloidal drops is examined using a fluorescence microscope and a synchronized side-view camera. The results show that the relaxation time of the water–air interface of the merged drop is shorter than that of a single drop impacting on a dry surface. It is also found that both morphology and particle distribution uniformity of the deposit change significantly with varying jetting delay and spatial spacing between two drops. As the drop spacing increases while keeping jetting delay constant, the circularity of the coalesced drop reduces. For the regime where the time scale for drop evaporation is comparable with the relaxation time scale for two drops to completely coalesce, the capillary flow induced by the local curvature variation of the air–water interface redistributes particles inside a merged drop, causing suppression of the coffee-ring effect for the case of a high jetting frequency while resulting in a region of particle accumulation in the middle of the merged drop at a low jetting frequency. By tuning the interplay of wetting, evaporation, capillary relaxation, and particle assembly, the deposition morphology of consecutively printed colloidal drops can be controlled.

Monday, November 19, 2012 3:35PM - 5:32PM –

Session L9 Interfacial/Thin Film Instability V: Evaporation 25B - Chair: Omar Matar, Imperial College London

3:35PM L9.00001 Stability of evaporating liquid layer with insoluble surfactant[1] , ALEXANDER MIKISHEV, Physics Department, Sam Houston State University, Huntsville, TX 77341, ALEXANDER NEPOMNYASHCHY, Mathematics Department, Technion, Haifa 32000, Israel — A horizontally infinite layer of incompressible Newtonian liquid with insoluble surfactant on the deformable surface is considered. The evaporation process is described by 2D one-side model. Therefore, the assumptions of small density, small viscosity and small thermal conductivity of the gaseous phase compared to the same properties of the liquid phase are accepted. Surface tension of the liquid-vapor surface linearly depends on temperature and concentration of surfactant. Using the long-wave approximation and assumption of slow time evolution the system of nonlinear equations is obtained. That system describes the spatiotemporal behavior of the layer interface and the field of the surfactant concentration. The equations retain all relevant physical effects which take place in the system. Linear stability analysis of the base state in the case of quasi-equilibrium evaporation, when the interfacial temperature equals the saturation one, is performed.

[1]The research was partially supported by the European Union via the FP7 Marie Curie scheme [Grant No.PITN-GA-23008-214919 (MULTIFLOW)].

3:48PM L9.00002 Influence of evaporation on a thin binary liquid film flowing down a heated inclined plate[1] , JUTHAMAS KAMRAK, BENOIT SCHEID, PIERRE COLINET, Universite Libre de Bruxelles-TIPs (Transfers, Interfaces and Processes) — We investigate the evolution of a two-component liquid film (here consisting of glycerine in water) falling down a heated plate, while water evaporates (glycerine is assumed to be non-volatile). The liquid phase is separated from pure water vapour by a deformable interface. We study the influence of both heat and mass transfer on the evolution of the liquid film. The temperature and concentration variations due to the evaporation of the solvent induce thermal and solutal Marangoni stresses on the free surface, thus affecting the evolution of the film. The mathematical model is developed by combining the lubrication theory with a weighted residuals approach. We obtain a set of coupled equations for the evolution of the film thickness, the velocity, the temperature and the concentration fields, at first-order. Stationary solutions are then calculated for different control parameters and show an intricate dependence of the different variables in the transition region where the evaporation flux reaches its maximum.

[1]Supported by FRIA, by the Marie Curie MULTIFLOW Network, and by FRS-FNRS.

4:01PM L9.00003 Solutal Marangoni instability in a binary liquid layer evaporating into air: the importance of transients in the gas for highly unstable cases , HATIM MACHRAFI, Universite de Liege, ALEXEY REDNIKOV, PIERRE COLINET, Universite Libre de Bruxelles, PIERRE DAUBY, Universite de Liege — This study considers an evaporating horizontal binary-liquid layer (aqueous solution of ethanol; mass fraction 0.1) in contact with air with an imposed transfer distance. Fully transient and horizontally homogeneous solutions for the reference state are first calculated. Then, the linear stability of these solutions is studied using the frozen-time approach. Solutal and thermal Rayleigh-Bénard-Marangoni instabilities are taken into account together with the Soret effect, although the solutal Marangoni mechanism appeared to be the most important one. Considering several gas-to-liquid thickness ratios (H), we calculate the critical times for the instability onset in a liquid layer of a given thickness. We also uncover the minimum liquid thicknesses under which no instability can ever occur. We subsequently observe that two distinctly different types of minimum thicknesses exist depending on H, examining each one of them. Then a closed-form analysis of the instability at small times has been developed. Finally, it has also been observed that, regardless of the gas-to-liquid thickness ratio, an asymptotic value of the critical time exists as the liquid layer increases, this critical time being approximately 1 μs.

4:14PM L9.00004 Experimental studies of a volatile simple fluid subject to a horizontal temperature gradient[1] , YAOFA LI, BENJAMIN CHAN, MINAMI YODA, Georgia Institute of Technology — Our fundamental understanding of transport in a nonisothermal liquid layer in the presence of evaporation and condensation is limited. Particle-image velocimetry (PIV) was used to measure 2D-2C liquid-phase velocity fields in layers of water and low-viscosity volatile silicone oil with an average depth of a few mm subject to a temperature difference of about 10 $^\circ$C over a horizontal distance of $\sim$5 cm. Two-color laser-induced fluorescence was also used to measure liquid-phase temperature fields in water. The liquid layers, which were confined in a 1 cm deep rectangular test cell, were studied under both ambient air and in equilibrium with their vapor. Given that thermocapillarity due to the changes in temperature associated with phase change at the free surface can be significant in these flows, results for water, and water with surfactant concentrations below and above the critical micellar concentration are compared. The experimental data for both fluids are also compared with numerical simulations.

[1]Supported by ONR.

4:27PM L9.00005 Numerical studies of a volatile simple fluid subject to a horizontal temperature gradient[1] , TONGRAN QIN, ROMAN GRIGORIEV, Georgia Institute of Technology — Rayleigh-Bénard and Marangoni convection in a layer of a homogeneous fluid with a free surface in the absence of phase change is a classic–and extensively studied–problem of fluid mechanics. The balance between the buoyancy and thermocapilly forces, however, depends, in a rather delicate manner, on the conditions of the experiment. Phase change, in particular, has a major effect on convection. Significant latent heat absorbed or released at the free surface as a result of evaporation or condensation can dramatically alter the interfacial temperature, and hence, thermocapillary stresses. This talk will use numerical simulations to illustrate how the phase change rates and the interfacial temperature distribution depend on two key factors: the accommodation coefficient of the working fluid and the presence of non-condensable gases, such as air. We will also compare numerical results with experiments.

[1]This work has been supported by ONR

4:40PM L9.00006 Experimental studies of volatile binary fluids subject to a horizontal temperature gradient[1] , MINAMI YODA, YAOFA LI, BENJAMIN CHAN, Georgia Institute of Technology — Convection in a binary fluid with a free surface in the presence of evaporation and condensation is a complex and poorly understood problem. In a two-component fluid where one component is more volatile and has a lower surface tension σ than the other, the surface tension of the mixture increases as temperature increases due to solutocapillary effects. In contrast, σ decreases as temperature increases in a simple fluid due to thermocapillary effects. The dynamics of the flow in O(1 mm) layers of dilute water-methanol mixtures driven by a temperature difference of about 10 $^\circ$C over a horizontal distance of $\sim$5 cm was studied using 2D-2C particle-image velocimetry (PIV). The liquid layers, which are confined in a 1 cm deep rectangular test cell, are studied under both ambient air and in equilibrium with their vapor(s). The experimental data for these binary fluids are compared with numerical simulations.

[1]Supported by ONR

4:53PM L9.00007 Numerical studies of a volatile binary fluid subject to a horizontal temperature gradient[1] , ROMAN GRIGORIEV, TONGRAN QIN, Georgia Institute of Technology — Convection in a volatile binary fluid with a free surface is an extremely rich, and still a relatively poorly understood, problem. The richness comes from the interplay of three different forces: buoyancy, thermocapillarity, and solutocapillarity. While solutocapillarity is typically associated with the Soret effect, in the presence of phase change this force arises primarily as a result of differential evaporation (or condensation) of the two components of the binary mixture. In this talk we will discuss some interesting manifestations of solutocapillarity, such as the instability leading to the emergence of traveling waves analogous to hydrothermal waves is systems driven by thermocapillarity. Comparison of numerical results with experiments will also be provided.

[1]This work has been supported by ONR

5:06PM L9.00008 On phase change in thermocapillary flows[1] , PEDRO SAENZ, PRASHANT VALLURI, KHELLIL SEFIANE, University of Edinburgh, GEORGE KARAPETSAS, University of Thessaly, OMAR MATAR, Imperial College London — We present the findings from our 3D direct numerical study of thermocapillary flows undergoing phase change. A liquid-gas model with VOF interface-tracking technique is employed to investigate stable and unstable (hydrothermal waves) scenarios. The spatiotemporal evolution of the local evaporation flux is determined with the assumption that vapour phase just above interface is at a local thermodynamic equilibrium with the liquid phase below. The transient vapour distribution in the gas is also accounted for by means of the solution of an advection-diffusion equation. We calculate the resulting spatially non-uniform flux and illustrate its controlling mechanisms, which involve the Marangoni effect and non-uniform vapour-pressure distribution due to the externally-imposed thermal gradient. We also present the flux's non-linear evolution due to the transient liquid-level reduction and its stabilizing-destabilizing effect on the thermal and physical interface fluctuations. The oscillatory temperature- and vapour-fields in the gas, tightly coupled with advection rolls observed, are also shown.

[1]EPSRC DTA

5:19PM L9.00009 Evaporative Instability in Binary Mixtures[1] , RANGA NARAYANAN, ERDEM UGUZ, University of Florida — In this talk we depict the physics of evaporative convection for binary systems in the presence of surface tension gradient effects. Two results are of importance. The first is that a binary system, in the absence of gravity, can generate an instability only when heated from the vapor side. This is to be contrasted with the case of a single component where instability can occur only when heated from the liquid side. The second result is that a binary system, in the presence of gravity, will generate an instability when heated from either the vapor or the liquid side provided the heating is strong enough. In addition to these results we show the conditions at which interfacial patterns can occur.

[1]Support from NSF OISE 0968313, Partner Univ. Fund and a Chateaubriand Fellowship is acknowledged.

Monday, November 19, 2012 3:35PM - 5:45PM –

Session L10 Instability: Jets, Wakes and Shear Layers VI: Jets 25C - Chair: Ann Karagozian, University of California, Los Angeles

3:35PM L10.00001 Experimental Study of Axial Forcing on a Swirling Jet , AMY MCCLENEY, PHILIPPE BARDET, The George Washinton University — An experimental swirling jet is created by independently controlling the axial and angular momentum injection with the resulting water jet discharging freely into a large tank. The jet is excited to enhance mixing in low to high swirl number regimes. Axial forcing on the jet is imposed for Strouhal number ranging from 0 to 15, Reynolds number from 1,000 to 10,000, and Swirl number from 0 to 1.3, where limited experimental data exists. The forcing amplitude is changed from 5 to 20 percent of the axial flow rate, while the azimuthal momentum injection stays constant; the resulting forcing creates a jet with varying swirl number. Swirling jets enhance the growth and mixing of fluids compared to non-swirling jets; this can lead to shorter combustion chambers and increased combustion efficiency. This mixing can be enhanced further by forcing natural instabilities in the jet. These imposed disturbances are either axial, which generates vortex rings, or angular, which create more complex structures. Past research involving forcing with swirling jets resulted in limited findings due to the concentration of forcing in either axial or angular directions. The flow structures of forced and steady jets are observed using PLIF through azimuthal dye injection.

3:48PM L10.00002 Unsteady Surface-Pressure Measurements and Time-Resolved Flow Visualization of a Normally and Obliquely Impinging Jet , MALEK AL-AWENI, AHMED NAGUIB, Michigan State University — In comparison to heat transfer and flow field data, there is little information regarding the space-time characteristics of the surface pressure fluctuations in impinging jets. Moreover, when available, such measurements are rarely accompanied by flow field information, making it difficult to identify the mechanisms/flow structures leading to pressure generation. The current work employs an axisymmetric air jet impinging on a flat wall to identify the flow features associated with significant wall-pressure signatures. To this end, unsteady surface-pressure measurements, using an embedded electret microphone array in the impingement plate, and time-resolved smoke-wire flow visualization are carried out for normal and oblique incidence of the jet. The results show the presence of two dominant modes of pressure oscillation, one is observed throughout the measurement domain and the other found primarily within the wall-jet zone. Links are drawn between these modes and the flow structures, yielding information useful for constructing structure-based models of the surface pressure in impinging jets.

4:01PM L10.00003 Convective and Absolute Instability of Liquid Jets under Gravity Effects , GHOBAD AMINI, MATTHIAS IHME, University of Michigan, Ann Arbor, ALI DOLATABADI, Concordia University — The break-up of liquid jets is of practical importance for several applications, including liquid-fuel-injection and ink-jet printing. In this work, the effect of gravity on the onset and growth rate of absolute and convective instabilities in liquid jets is studied. The mathematical problem is formulated in terms of quasi-one-dimensional equations, and the linearized stability equations are solved using a first-order perturbation method. An analytic form of the dispersion equation is derived, and the variation of the growth rate is investigated for a range of positive and negative Bond numbers, corresponding to downward-pointing and rising liquid jet. The critical Weber number, demarcating the transition between convective and absolute instability is determined as function of Reynolds and Froude numbers. Model-results for the limiting case of zero gravity are compared with classical results of Chandrasekhar and Leib & Goldstein, confirming the validity of this approach.

4:14PM L10.00004 ABSTRACT WITHDRAWN –

4:27PM L10.00005 Preferred modes in jets: comparison between different measures of the receptivity[1] , XAVIER GARNAUD, LUTZ LESSHAFFT, PETER J. SCHMID, PATRICK HUERRE, CNRS - Ecole Polytechnique — The response of jets to frequency forcing is usually measured experimentally in terms of the maximum amplitude of velocity fluctuations reached along the axis (Crow & Champagne (1971)). In the present work, the preferred mode of isothermal jets is discussed in terms of the linear flow response to time-harmonic forcing (Trefethen *et al.* (1993)). The optimal frequency response is computed for different choices of the objective functional: the usual energy (L^2) norm and the maximum amplitude over the entire domain (L^∞ norm). The relevance and limitations of the different objective functionals are critically analyzed. Although the dominant flow structures are robustly identified in all cases, the measure of the flow response in terms of the maximum amplitude does not suffer from the continually slow axial growth of low frequency perturbations.

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

4:40PM L10.00006 Structural Variation in Convectively and Absolutely Unstable Jets in Crossflow[1] , DANIEL GETSINGER, LEVON GEVORKYAN, OWEN SMITH, ANN KARAGOZIAN, University of California, Los Angeles — This experimental study explores the behavior of both unforced and acoustically forced variable density transverse jets, via acetone PLIF and stereo PIV measurements. Jets composed of mixtures of helium and nitrogen are injected normally from a converging nozzle into an air crossflow, for a range of jet-to-crossflow momentum flux ratios J and density ratios S. A recent study[2] determined, based on hotwire-based spectral characteristics and excitation response, that transverse jet shear layers transition to global instability in response to sufficient lowering of S (below 0.45-0.40) and/or sufficient lowering of J (below 10). The changes in flow structure during such transitions are documented in the present study, where alterations in the transverse jet's vorticity field, cross-sectional symmetry or asymmetry, and dynamic flow features are affected. Both absolutely unstable transverse jets and forced transverse jets are observed to have more symmetric cross-sections than those for convectively unstable jets at higher J values, the latter of which also can exhibit the presence of tertiary vortex structures first identified by Kuzo.[3]

[1]Supported by NSF grant CBET-1133015 & AFOSR grant A001768901.
[2]Getsinger, et al., **Expts in Fluids**, 2012
[3]Kuzo, D., Ph.D. thesis, Caltech, 1995

4:53PM L10.00007 Mixing Characteristics of Convectively and Absolutely Unstable Jets in Crossflow[1] , LEVON GEVORKYAN, DANIEL GETSINGER, OWEN SMITH, ANN KARAGOZIAN, University of California, Los Angeles — This experimental study explores the mixing characteristics of both unforced and acoustically forced variable density transverse jets via acetone PLIF measurements. A range of jet-to-crossflow momentum flux ratios J and density ratios S are explored in this study, spanning conditions for which the jet shear layer transitions from being convectively to absolutely unstable. While there are clear differences in the flow structure among convectively unstable, absolutely unstable, and externally forced jets in crossflow, it is of interest in the present study to explore the implications of such differences for jet mixing. A range of metrics is used to quantify jet mixing, including jet centerline concentration decay and spread,[2] and the spatial evolution of jet cross-sectional Unmixedness, Spatial Mixing Deficiency, and Scale of Segregation.[3] It is found that the cross-sectional jet fluid distribution can be affected significantly by unforced shear layer stability characteristics as well as the nature of jet forcing, and hence can affect mixing in ways that are not evident from centerplane mixing metrics alone.

[1]Supported by NSF grant CBET-1133015 & AFOSR grant A001768901.
[2]Smith, S. H. and Mungal, M. G., **JFM**, 357, 83-122, 1998
[3]D. Bothe in **Micro and Macro Mixing**, 17-35, Springer 2010

5:06PM L10.00008 Effect of dilatant additive on stability of waterjet , NARIMAN ASHRAFI, Azad University — Effect of addition of dilatant cornstarch on the stability and precision enhancement of the abrasive waterjet is studied. It is shown that the normal stresses developed in the nonlinear viscoelastic additive remains substantially unchanged throughout effective length of the jet resulting in an almost completely prismatic jet, applicable for precision and straight machining. Furthermore, the jet becomes more stable upon increasing the cornstarch percentage. Clearly, there is restriction of the pump delivery upon adding the dilatant cornstarch. Different percentages of the additive are therefore examined. It is found that, a %22 additive results in the best performance based on the precision, required pump power and stability of the jet. Simulation of the problem is in good agreement with the experimental observations. The additive also appears to produce less friction with the surrounding air avoiding possible jet disintegration

5:19PM L10.00009 Vortex breakdown in a coaxial swirling jet with a density difference: an experimental study , AHMAD ADZLAN FADZLI BIN KHAIRI, Universiti Malaysia Sarawak, HIROSHI GOTODA, Ritsumeikan University — The vortex breakdown (VB) in a coaxial swirling jet with a density difference has been experimentally investigated in this work, focusing on how the Reynolds number of the outer jet affects the presence and height of the stagnation point in the swirling jet undergoing VB. When the Reynolds number of the outer jet is increased, the degree of flow divergence of both air and CO_2 jets decrease, with the degree of flow divergence of the air jet decreasing more compared to the CO_2 jet at the same Reynolds number of the outer jet. The stagnation point for the air jet was also found to disappear more easily compared to the CO_2 jet, and its height of the stagnation point increases more compared to that of the CO_2 jet. Investigation of the velocity field revealed that increasing the Reynolds number of the outer jet during VB induces a decrease in the gradient of the axial velocity distribution along the perpendicular line connecting the stagnation point and the $r = 0$ plane, causing the height of the stagnation point to increase. When the distribution no longer tends to zero, the stagnation point disappears. A physical model is considered in this presentation to explain these observations by estimating the momentum balance in the flow, based on a simplified Navier-Stokes equation (A. Adzlan, H. Gotoda, Chemical Engineering Science, 2012) – the first of its kind reported in the field of fluid dynamics.

5:32PM L10.00010 DNS of the transition from absolute to convective instability for transverse jets[1] , ERIN MUSSONI, PRAHLADH IYER, KRISHNAN MAHESH, 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. Behavior of fluid flow in the near field, such as the recirculation region inside the nozzle for low jet velocity ratios, is discussed to examine the stability transition with increasing jet velocity ratio. Global stability analysis is used to further explore the absolute to convective instability transition.

[1]Supported by NSF

Monday, November 19, 2012 3:35PM - 5:45PM –
Session L11 Bubbles IV and Drug Delivery 26A - Chair: Kausik Sarkar, George Washington University

3:35PM L11.00001 Simulation of Magnetic Particles in the Bloodstream for Magnetic Drug Targeting Applications , ERICA CHERRY, Student at Stanford University, JOHN EATON, Professor at Stanford University — Magnetic Drug Targeting (MDT) is a promising new idea for treatment of cancer and other well-localized diseases. An ideal MDT treatment would involve chemically binding the drug particles to magnetic particles, injecting them into the body, and using external magnetic fields to steer the particles towards or hold them near areas of diseased tissue not accessible via injection. However, it would be difficult to implement efficient MDT treatments because we know little about how magnetic particles interact with blood flow. With the goal of understanding these dynamics, a simulation of blood flow containing magnetic particles was performed. The particles were subject to a variety of forces such as gravity, externally-applied magnetic force, and inter-particle magnetic force. A separate simulation was performed to determine how the magnetic particle dispersion coefficient varied with flow properties such as shear and erythrocyte content, and the results of the dispersion simulation were used in the main simulation. Results from these simulations will be presented and used to draw conclusions about the technology required for a successful MDT treatment.

3:48PM L11.00002 Highly-focused high-speed impact on soft material: Application for needle-free injection device , YOSHIYUKI TAGAWA, NIKOLAI OUDALOV, CLAAS WILLEM VISSER, CHAO SUN, DETLEF LOHSE, University of Twente — The development of needle-free drug injection systems is of great importance to global healthcare. Existing methods use diffusive jets, which suffer from insufficient penetration into the skin. We established a novel method of creating microjets with a very sharp geometry and controlled velocities even for supersonic speeds up to 850 m/s. In this presentation we demonstrate that it is possible to penetrate human skin using these jets and in this way deliver liquid substances to the human body. The penetration depth is much deeper than those of conventional methods. Further penetration dynamics is studied through experiments performed on gelatin mixtures. A model based on Stokes-like drag is proposed to predict the depth of the penetration.

4:01PM L11.00003 Mapping the acoustic scattering behavior of spherical microbubble clouds , MIGUEL A. PARRALES, JUAN M. FERNÁNDEZ, MIGUEL PEREZ-SABORID, Dept. Ingenieria Aeroespacial y Mecanica de Fluidos, Universidad de Sevilla, Spain — Sound scattering and acoustic propagation through bubbly liquids have been studied deeply in the last decades. The main reason for these studies was to explain and analyze the high impact of gas bubbles on sound propagation: the compressibility missmatch and the resonant behavior make the bubble a very efficient sound scatterer, changing appreciably the acoustic properties of the biphasic medium. Here we propose a numerical analysis, based on the self-consistant multiple scattering approach, to compute the linear acoustic response of spherical microbubble clouds while excited by an external ultrasonic wave. The calculations have been done for a wide range of the cloud void fraction β. By varying the excitation frequency ω_o, we are able to map the total scattering intensity from the cloud in a $(\beta - \omega_o)$ phase space. The localization of the collective resonant modes on this map finally reveals the different scattering regimes. Furthermore, the total pressure field is obtained both inside and outside the cloud, being possible to visualize the acoustic wave propagation in each scattering regime.

4:14PM L11.00004 Acoustic Excitation of a Micro-bubble Inside a Rigid Tube , ADNAN QAMAR, RAVI SAMTANEY, Physical Sciences and Engineering Division, King Abdullah University of Science and Technology, Thuwal, KSA — A theoretical model for acoustic excitation of a single micro-bubble inside a rigid tube is proposed in the present work. The model is derived from the reduced Navier-Stokes equations and by utilizing Poiseuille pipe flow theory. Wall Frictional losses induced due to fluid motion by the bubble oscillation in response to the acoustic perturbation are taken into account. The proposed model is not a variant of conventional Rayleigh-Plesset (RP) equation and is principally a super-set of all the conventional RP models. The model is first of its kind, which relates the bubble dynamics with the tube geometric and acoustic parameters in a consistent manner. Model predicts bubble oscillation dynamics as well as bubble fragmentation quite well when compared to the available experimental data. Results are computed for three tube diameters of 200, 100 and 12 microns with two initial bubble radiuses of 1.5 and 2 microns. The response of micro-bubble is highly non-linear with the driving acoustic frequency. Bubble response for low acoustic peak negative pressure (PNP) is linear, whereas as the PNP is increase nonlinearity are manifested and eventually bubble fragmentation takes place. For fixed acoustic parameters, an exponential decay in bubble response is observed as the tube length is increased. For very small tube diameters, the predictions are damped, suggesting the breakdown of the inherent model assumptions for these cases.

4:27PM L11.00005 Stably Levitated Large Bubbles in Vertically Vibrating Liquids , TIMOTHY O'HERN, BION SHELDEN, LOUIS ROMERO, JOHN TORCZYNSKI, Sandia National Laboratories — Vertical vibration of a liquid can cause small gas bubbles to move downward against the buoyancy force. Downward bubble motion is caused by the oscillating bubble volume (induced by the oscillating pressure field) interacting with the bubble drag force. The volume-drag asymmetry and the oscillating pressure gradient produce net downward bubble motion analogous to that caused by the Bjerknes force in high-frequency vibrations. Low-frequency (below 300 Hz) experiments demonstrate downward bubble motion over a range of vibration conditions, liquid properties, and pressure in the air above the free surface. Small bubbles deep in a quasi-two-dimensional test cell usually coalesce to form a much larger bubble that is stably levitated well below the free surface. The size and position of this levitated bubble can be controlled by varying the vibration conditions. 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.

4:40PM L11.00006 Nonlinear dynamics of PLA (poly-lactic acid) encapsulated ultrasound contrast microbubbles[1] , SHIRSHENDU PAUL, University of Delaware, KAUSIK SARKAR, George Washington University, MARGARET WHEATLEY, Drexel University — The presence of the stabilizing encapsulation in microbubble based ultrasound contrast agents (UCAs) has critical effects on their acoustic properties. Biodegradable polymers like poly-lactic acid (PLA) hold promises to provide better stability and control over properties. Here, we report determination of interfacial rheological properties of PLA microbubbles using *in vitro* experiments and investigation of their non-linear scattering response. The average bubble size measured using DLS is 1.8 μm. However, the attenuation measured through a suspension of PLA bubbles shows a peak between 2.5-3.2 MHz, much smaller than the resonance frequency of a free bubble of similar size. This observation is explained by an extremely low surface elasticity (0.02-0.06 N/m) and the polydispersity of the bubble population. The estimated properties lead to an excellent agreement between model prediction and the experimentally measured response (up to 30 dB enhancement of fundamental response). Subharmonic threshold prediction is shown to be critically dependent on the bubble size distribution.

[1]Partially supported by NSF

4:53PM L11.00007 Microbubbles as drug-delivery vectors: steering ultrasound contrast agents in arterial flow using the Bjerknes force[1] , ALBERTO ALISEDA, ALICIA CLARK, University of Washington Department of Mechanical Engineering — Micron-sized coated microbubbles, commonly referred to as ultrasound contrast agents (UCAs), have been identified as potential targeted drug delivery vectors with applications in cancer chemotherapy and thrombolysis. The Bjerknes force, produced by the fluctuating pressure field created by the ultrasound waves acting on the oscillating bubble with a phase lag induced by the liquid's inertia and viscosity, can be used to direct the microbubbles to specific targeted areas in the circulatory system. While this phenomenon is well understood in a quiescent fluid, we need a better understanding of the dynamics of microbubbles in the complex pulsatile flow found in the human circulatory system. The non-linear interactions of ultrasound volume oscillations and flow-induced stresses are explored via high speed imaging of UCAs under in vitro flow that reproduces conditions in large arteries (relatively high Reynolds and Womersley numbers). This improved understanding will be used to manipulate and steer UCAs with ultrasound, in conjunction with hydrodynamic forces.

[1]NSF CAREER, NSF Graduate Research Fellowship

5:06PM L11.00008 The Short Time Scale Events of Acoustic Droplet Vaporization[1] , DAVID S. LI[2], OLIVER D. KRIPFGANS[3], J. BRIAN FOWLKES[4], JOSEPH L. BULL[5], University of Michigan — The conversion of a liquid microdroplets to gas bubbles initiated by an acoustic pulse, known as acoustic droplet vaporization (ADV), has been proposed as a method to selectively generate gas emboli for therapeutic purposes (gas embolotherapy), specifically for vascularized tumors. In this study we focused on the first 10 microseconds of the ADV process, namely the gas nucleation site formation and bubble evolution. BSA encapsulated dodecafluoropentane (CAS: 678–26–2) microdroplets were isolated at the bottom of a degassed water bath held at 37°C. Microdroplets, diameters ranging from 5-65 microns, were vaporized using a single pulse (4-16 cycles) from a 7.5 MHz focused single element transducer ranging from 2-5 MPa peak negative pressure and images of the vaporization process were recorded using an ultra-high speed camera (SIM802, Specialised Imaging Ltd). It was observed that typically two gas nuclei were formed in series with one another on axis with ultrasound pulse. However, relative positioning of the nucleation sites within the droplet depended on droplet diameter. Additionally, depending on acoustic parameters the bubble could deform into a toroidal shape. Such dynamics could suggest acoustic parameters that may result in tissue damage.

[1]This work is supported by NIH grant R01EB006476.
[2]Department of Biomedical Engineering
[3]Department of Radiology
[4]Department of Radiology
[5]Department of Biomedical Engineering

5:19PM L11.00009 An Experimental Review on Microbubble Generation to Be Used in Echo-PIV Method to Determine the Pipe Flow Velocity , ALINAGHI SALARI, MOHAMMAD BEHSHAD SHAFII, Sharif University of Technology, SHAPOOR SHIRANI, Tehran Heart Center — Today microbubbles are broadly used as ultrasound contrast agents. Flow Focusing (FF) in microchannels can pave the way for the generation of same size bubbles. Microbubbles can be used as tracers in the Echo Particle Image Velocimetry (Echo-PIV) method to determine the velocity profile in main body vessels such as carotid. In this paper we use a low-cost microchannel fabrication method for preparing microbubble contrast agents by using some surface active agents and a viscosity enhancing material to obtain appropriate microbubbles with desired lifetime and stability for any *in vitro* infusion for velocity measurement. All the five parameters that govern the bubble size extract and some efforts are done to achieve the smallest bubbles by adding suitable surfactant concentrations. By using these microbubbles for the Echo-PIV method, we experimentally determine the velocity field of two flow types, namely steady state and pulsatile pipe flows.

5:32PM L11.00010 Development of microbubbles generator using microchannel toward biomedical applications , HIRONOBU KAJI, REI MASUDA, KAZUHITO INOUE, The University of Tokyo, MITSUHISA ICHIYANAGI, Sophis University, IKUYA KINEFUCHI, SHU TAKAGI, YOICHIRO MATUMOTO, The University of Tokyo — Microbubbles have been already used as ultrasound contrast agents to visualize microcirculation system. They are also expected to be used for the drag delivery agent. For these bubbles, important requirements are their size and functionality such as carrying drugs and staying stability in vivo. Aiming at the development of microbubbles with the well-controlled size and functions, we have been developing a microbubble generation system using microchannels. Advantages of the method using microchannels are to generate small- and monodisperse-size microbubbles with the wide variety of choice in both liquid phase and gas phase and the capability of surface coating. In the present study, microbubbles are generated using T-junction type microchannel. We have designed the channel shape to reduce the bubble size. The improvement of the shape has enabled us to generate the smaller microbubble whose diameter is 6.1μm. Moreover, the effect of the viscosity in the liquid phase are investigated and it is confirmed that smaller bubbles are generated with the increase of viscosity. In addition, we have developed a new type microchannel for the surface coating of a microbubble. The results will be discussed in the presentation.

Monday, November 19, 2012 3:35PM - 5:45PM –

Session L12 Vortex VI 26B - Chair: Jochen Kriegseis, University of Calgary

3:35PM L12.00001 Flow Structure on a Delta Wing of Moderate Sweep Angle During and After Pitch-Up Maneuver , ALPER CELIK, ILHAN OZTURK, HABIB CAN TUNC, MEHMET METIN YAVUZ, Middle East Technical University — The flow structure over a moderate sweep angle delta wing is investigated during and after the pitch-up maneuver and compared to the corresponding stationary wing results. The effects of pitch-up rate and Reynolds number on flow structures and their transformations are also studied. Dye visualization is used for qualitative studies and particle image velocimetry is used for quantitative analysis. At early stages of the maneuver the transformation of flow is initiated by the intertwinement of the dual vortex structure.Increasing the angle of attack results in disappearance of the vortex located closest to the leading-edge of the wing which results in a single, large scale, leading-edge vortex that undergoes a distinctive form of breakdown. It is found that the motion of the wing creates a significant time lag on the development of the flow pattern, when compared to stationary wing. In the relaxation period, the vorticity concentrations become more diffuse and elongated as they move towards the plane of symmetry, and away from the surface of the wing. All these features and transformations occur irrespective of values of pitch rates and Reynold's number. On the other hand, it is seen that the lag of flow pattern is a function of pitch rate and Reynold's number.

3:48PM L12.00002 The suppression of wing rock of wing/body configurations[1] , XUEYING DENG, Retired from BUAA, YANKUI WANG, Professor of BUAA, MINISTRY-OF-EDUCATION KEY LABORATORY OF FLUID MECHANICS, BEIJING UNIVERSITY OF AERONAUTICS AND ASTRO TEAM — The suppression and flow mechanism of wing rock for wing/body configurations with high and low swept wing at high angles of attack have been studied. The results of low swept wing model reveal that wing rock patterns are dominated by body asymmetric vortices which are strongly dependent on circumferential location θ of micro-perturbation on the body tip. There are three types of free roll patterns: limit cycle at θ=0° or 180°; irregular oscillation at θ=0° or 270°; tiny roll pattern, if θ is at the other positions. For low swept model a technique of suppressing wing rock by rotating tip perturbation was developed with higher frequency than one of free roll motion. For the model with high swept wing another technique of suppressing wing rock was developed, where two pairs of asymmetric vortices (one pair from body and the other from high swept wing) will dominate the wing rock. If two pairs of asymmetric vortices are in phase coincidence, it makes wing rock stronger while two pairs of vortices are in phase reversal the wing rock becomes weaker. Based on this fact the perturbation at body tip can be adjusted to suppress the wing rock.

[1]The project was supported by the National Natural Science Foundation of China (11172030).

4:01PM L12.00003 Vortex Interactions on Plunging Airfoil and Wings , AZAR ESLAM PANAH, JAMES BUCHHOLZ, University of Iowa — The development of robust qualitative and quantitative models for the vorticity fields generated by oscillating foils and wings can provide a framework in which to understand flow interactions within groups of unsteady lifting bodies (e.g. shoals of birds, fish, MAV's), and inform low-order aerodynamic models. In the present experimental study, the flow fields generated by a plunging flat-plate airfoil and finite-aspect-ratio wing are characterized in terms of vortex topology, and circulation at Re=10,000. Strouhal numbers (St=fA/U) between 0.1 and 0.6 are investigated for plunge amplitudes of h_o/c = 0.2, 0.3, and 0.4, resulting in reduced frequencies (k=π fc/U) between 0.39 and 4.71. For the nominally two-dimensional airfoil, the number of discrete vortex structures shed from the trailing edge, and the trajectory of the leading edge vortex (LEV) and its interaction with trailing edge vortex (TEV) are found to be primarily governed by k; however, for St>0.4, the role of St on these phenomena increases. Likewise, circulation of the TEV exhibits a dependence on k; however, the circulation of the LEV depends primarily on St. The growth and ultimate strength of the LEV depends strongly on its interaction with the body; in particular, with a region of opposite-sign vorticity generated on the surface of the body due to the influence of the LEV. In the finite-aspect-ratio case, spanwise flow is also a significant factor. The roles of these phenomena on vortex evolution and strength will be discussed in detail.

4:14PM L12.00004 Aircraft wake two-vortex system at turbulent equilibrium , GREGOIRE WINCKELMANS, IVAN DE VISSCHER, Universite catholique de Louvain (UCL), Institute of Mechanics, Materials and Civil Engineering (iMMC), LAURENT BRICTEUX, University of Mons (UMons), Fluids and Machines Dept. — We consider a two-vortex system (2VS) started from a 2-D initial condition of given energy (two opposite sign vortices, each with an algebraic circulation profile and with a relatively tight core, r_c, compared to the distance, b_0, center to center: $r_c/b_0 = 0.05$). The 2VS is submitted to a very weak and realistic atmospheric turbulence background (of energy < 0.01 that of the 2VS) so that it is excited to go unstable. The flow then generates, by non-linear interactions, instabilities and much more turbulence and eventually reaches a statistical equilibrium: a 2VS still with tight cores, with significant turbulence in the vortex oval, yet still laminar in the inner part of the cores, and which slowly decays in time. This state of equilibrium is quite universal (as confirmed by various sensitivity analyses). It is then of great importance to the physics and modeling of fully formed aircraft wake vortices and is characterized: spectrum, vorticity field, circulation profile and core size of the vortices in cross-planes and for the mean (i.e., longitudinal average). The two-scales Proctor-Winckelmans profile model is also compared to the data: it fits well the inner part $0 < r/b_0 < 0.04$ and the outer part $0.16 < r/b_0 < 0.5$ of the profile, but is still poor in between.

4:27PM L12.00005 Force Production from Near- and Far-Field Vortices on Flapping Wings , HUI WAN, ZONGXIAN LIANG, HAIBO DONG, Wright State University, FLOW SIMULATION AND RESEARCH GROUP TEAM — Aerodynamic performance is closely correlated with the vortex formation and vortex structure of a flapping plate. The aerodynamic forces experienced the plate can be obtained from the surface integration of the pressure and shear stress distribution, or alternatively from the time rate of change of the vortex moment in the flow field Direct Numerical Simulation (DNS) is first conducted to generate the flow field information including both velocity and vorticity. Then the aerodynamic forces obtained from the above two methods are calculated and compared. The roles of near-field and far-field vortex structures on force generation are studied. Both two-dimensional plate and low aspect-ratio plate will be discussed. This work is supported by NSF CBET-1055949.

4:40PM L12.00006 Modeling the dynamics of four-vortex bluff body wakes , SAIKAT BASU, MARK STREMLER, Virginia Tech, TEIS SCHNIPPER, ANDERS ANDERSEN, Technical University of Denmark — Vortex-shedding bluff bodies frequently generate wakes consisting of regular groupings of four vortices. We model such wakes as an integrable two-dimensional Hamiltonian system of point vortices by assuming spatial symmetries based on the vortex arrangements observed in experiments. The model demonstrates a number of dynamic modes that we classify using a bifurcation analysis of the phase space topology. In contrast to the standard von Karman street, very few initial conditions lead to relative equilibria in which the vortex configuration moves with invariant size and shape. Scaled comparisons of the model with experiments conducted in a flowing soap film support the model results. The richness of the results reveals a variety of patterns that can occur within the class of '2P' wakes. Following the approach of von Karman, we estimate the drag and lift forces exerted by such wakes on a bluff body.

4:53PM L12.00007 Effect of Relative Submergence on the Flow Structure in the Wake of Wall-Mounted Spherical Obstacle[1] , SEYED HAJIMIRZAIE, ACHILLEAS TSAKIRIS, JAMES BUCHHOLZ, ATHANASIOS PAPANICOLAOU, The University of Iowa — To understand the flow around submerged boulders in open channels, a study of a spherical obstacle on a rough bed in shallow open channel is conducted. In steep mountain streams, relative submergence (d/H, d being flow depth and H obstacle height) is introduced. In this study, through the use of PIV and thermal anemometry, the flow field surrounding a wall-mounted sphere with diameter D=5cm in two different relative submergences has been investigated on a smooth plate as a boundary layer (BL) as well as on a rough bed in shallow open channel flow. Flow patterns (velocity, vorticity) in the streamwise symmetry plane are different between the open channel and BL flow. Streamwise features are observed in the mean wakes of the sphere. In high relative submergence, an upwash (base structure) is observed in the wake of sphere in open channel flow but not in BL tests. The horseshoe vortex is not observed in the wake of the sphere in either case. Dye visualization, spectral analysis and cross-correlation show in d/H$\leq$1, the wake in both cases has the appearance of a Karman vortex street, while in d/H $>$ 1, the wakes are quite symmetric. A weak dominant shedding frequency is observed in BL experiments with a Strouhal number of St=fsD/U$\approx$0.35.

[1]This work is supported by NSF grant number CBET-1033732.

5:06PM L12.00008 Wake dynamics and hydrodynamic forces on a perforated circular plate in cross-flow[1] , FRANCISCO HUERA-HUARTE[2], Universitat Rovira i Virgili — The cross-flow past a perforated plate is known to become steady, if certain critical porosity or number of holes is imposed to the plate. This happens because the air bleed in the near wake, disrupts the vortex street formation behind the plate, and leads to suppression of the near wake shear layer interaction, forcing the instabilities to take place further away from the disk. This phenomenon is accompanied by a drag reduction. It is not clear however, what is the effect of the porosity distribution used in the plate, neither the effect of the angle of attack on the wake dynamics and the force coefficients. The experimental apparatus consists of an acrylic model in which different number and configuration of holes can be used. The disk hangs upside down from a 2-axis balance, in a way that it is being exposed to a uniform water current generated in a free surface channel. Angles of attack, porosity and its distribution on the disk, can be easily changed. Measurements of force coefficients for different angles of attack, and porosities have been taken. Digital Particle Image Velocimetry (DPIV) has been used to quantify the wake and to investigate the flow structures past the disk.

[1]Funding provided by the Spanish Ministry of Science through grant DPI2009-07104 is acknowledged.

[2]Associate Prof. Dept. Mechanical Engineering

5:19PM L12.00009 Influence of diameter ratio and aspect ratio on wake development of a dual step cylinder[1] , SERHIY YARUSEVYCH, CHRIS MORTON, University of Waterloo — A dual step cylinder is composed of a large diameter cylinder (D) attached to the mid-span of a small diameter cylinder (d). In a uniform cross flow, vortex shedding occurs from the small cylinder, above and below the large cylinder. The characteristics of the shed vortices are similar to those found in the wake of a uniform circular cylinder. However, wake characteristics of the large cylinder are influenced significantly by the geometric parameters of the model, namely, the ratio between the large and small cylinder diameters (D/d) and the large cylinder aspect ratio (L/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 have been completed in a water flume facility employing Laser Doppler Velocimetry (LDV) and Particle Image Velocimetry (PIV) systems, as well as hydrogen bubble flow visualization. The results show that the following three distinct large cylinder wake topologies can be observed for the investigated ranges of L/D and D/d: (i) shedding of uniform spanwise vortices, (ii) shedding of highly deformed three-dimensional vortices, and (iii) no distinct vortex shedding. Complex vortex interactions taking place in the wake of the large cylinder are investigated for the identified flow regimes.

[1]The authors gratefully acknowledge the Natural Sciences and Engineering Research Council of Canada (NSERC) for funding of this work.

5:32PM L12.00010 ABSTRACT WITHDRAWN –

Monday, November 19, 2012 3:35PM - 5:32PM –

Session L13 Rayleigh-Taylor/Richtmyer-Meshkov Instability 27A - Chair: Jeffrey Jacobs, University of Arizona

3:35PM L13.00001 Turbulent mixing driven by a spherical converging shock , M. LOMBARDINI, D.I. PULLIN, D.I. MEIRON, California Institute of Technology, R.A. GORE, Los Alamos National Laboratory — We present recent results from large-eddy simulations of the mixing induced at a perturbed spherical density interface initially impacted by a spherically convergent shock wave of Mach number $\simeq 1.2$ at impact, and then re-shocked in the expansive phase. Two configurations are compared: i) air inside and SF_6 (five times denser than air) outside, i.e. heavy–light configuration; ii) SF_6 inside and air outside, or light–heavy configuration. From data interpolated over spherical surfaces, we compute various power spectra as well as extensive surface-averaged statistics involved in the budget of turbulent kinetic energy and enstrophy density.

3:48PM L13.00002 Experiments on the Rayleigh-Taylor instability of gas-gas interfaces accelerated by an expansion wave , 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 gases at matched volumetric flow rates. The interface is accelerated by an expansion wave generated by the rupturing of a diaphragm separating the heavy gas from a vacuum tank evacuated to approximately 0.1atm. The expansion wave generates a very high, O(1e3g0), but non-constant acceleration on the interface causing the Rayleigh-Taylor instability to develop. Shadowgraphy is employed to visualize the instability using two sets of three 200 mm diameter f/6.0 parabolic mirrors and three CMOS cameras operating at 10kHz with exposure times of 1e-6s. Planar Mie scattering is also employed using a planar laser sheet generated at the top of the apparatus which illuminates smoke particles seeded in the heavy gas. The scattered light is then imaged using three CMOS cameras operating at 10kHz. Experiments are shown in which a random perturbation is introduced by vertically oscillating the fluid interface to produce Faraday waves.

4:01PM L13.00003 Progress on experimental investigation of RT instability at high Atwood numbers , BHANESH AKULA, DEVESH RANJAN, Texas A&M University — The new multi layer experimental facility at Texas A&M University can be used to study the mixing between two or more gas streams (separated by partitions initially) with different densities and velocities. This is a convective type system similar to the gas channel facility that was used to study RT mixing. This new suction-type multi layer facility has a test section double the size of the gas channel which will enable measurements up to Reynolds number of 30000. For the present study, this facility is used to study the Rayleigh-Taylor mixing between Air and Air-Helium mixtures at Atwood numbers greater than 0.5. Different diagnostics including Simultaneous PIV-PLIF and backlight imaging are used to obtain field wise measurement of velocities and densities as well as mixing width and its growth rate. The parameters obtained from these measurements including molecular mixing parameter θ, turbulent quantities such as mean fluctuation of streamwise and cross stream velocities are presented.

4:14PM L13.00004 Miscible and immiscible, forced and unforced experiments on the Rayleigh-Taylor instability , MICHAEL ROBERTS, MATTHEW MOKLER, JEFFREY JACOBS, University of Arizona — Experiments are presented in which an incompressible system of two liquids is accelerated to produce the Rayleigh-Taylor instability. In these experiments, the initially stable, stratified liquid combination is accelerated downward on a vertical rail system in one of two experimental apparatuses: an apparatus in which a system of weights and pulleys accelerates the liquid filled tank, or a new apparatus which uses linear induction motors to accelerate the tank to produce much greater acceleration levels. Both miscible and immiscible liquid combinations are used. In both apparatuses the resulting fluid flows are visualized with backlit imaging using LED backlights in conjunction with monochrome high-speed video cameras, both of which travel with the moving fluid filled containers. Initial perturbations are either unforced and allowed to progress from background noise or forced by vertically oscillating the liquid combination to produce parametric internal waves. The mixing layer growth rate α is determined for all cases and compared to numerical simulations and past experiments.

4:27PM L13.00005 Effects of Initial Conditions on Rayleigh-Taylor Instability in Elastic-Plastic Materials[1] , PAMELA ROACH, Missouri University of Science & Technology, Rolla, MO, ARINDAM BANERJEE, Lehigh University, Bethlehem, PA — In contrast to Newtonian fluids, experimental study of Rayleigh Taylor instability (RTI) in accelerated solids is traditionally hindered by difficulty to measure material properties and exceedingly small time scales of the processes. When an elastic-plastic solid is accelerated due to a density gradient, the instability is dependent on the material's mechanical strength, initial conditions, and acceleration that drive the instability. RTI in solids is observed in supernovas, explosive welding, and inertial confinement fusion. A novel experimental technique is used to study the effects of initial conditions and variable accelerations on the growth and instability in 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 are analyzed and effects of two-dimensional versus three-dimensional interfaces are discussed. Furthermore, the instability threshold and stable elastic and plastic regions are investigated by controlling the acceleration. The instability threshold and perturbation growth rate are compared to linear analysis of incompressible RTI.

[1]The authors acknowledge financial support through DOE-LANL subcontract # 173667-1.

4:40PM L13.00006 Compressibility and Stratification Effects on Single-Mode Rayleigh-Taylor Instability , SCOTT RECKINGER, University of Colorado Boulder, DANIEL LIVESCU, Los Alamos National Laboratory, OLEG VASILYEV, University of Colorado Boulder — Simulations of single-mode compressible Rayleigh-Taylor instability (RTI) are performed using the Adaptive Wavelet Collocation Method (AWCM). Due to the physics-based adaptivity and direct error control of the method, AWCM is ideal for resolving the wide range of scales present in RTI growth. AWCM is used in conjunction with non-reflecting boundary conditions developed for highly stratified systems. This combination allows for extremely long domains, which is necessary for observing the late time growth of compressible RTI. The background state consists of two diffusively mixed stratified fluids of differing molar masses. Of interest are the compressibility effects on the departure time from the linear growth, the onset of strong non-linear interactions, and the late-time behavior of the fluid structures. For initial conditions corresponding to thermal equilibrium, the background stratification suppresses the instability growth when the molar masses are similar. A reversal in this monotonic behavior is observed for large molar mass differences, when stratification acts to enhance the bubble growth. The effects of the background stratification on the late-time vorticity generation and the associated induced velocities are also investigated.

4:53PM L13.00007 Nonlinear evolution of Richtmyer-Meshkov and Rayleigh-Taylor instabilities in a domain of a finite size[1] , A. QAMAR, S.I. ABARZHI, University of Chicago, Chicago, IL, USA — We developed theoretical analysis to systematically study the nonlinear evolution of Richtmyer-Meshkov and Rayleigh-Taylor instability in a domain of a finite size. Fluids have either similar or contrasting densities, and acceleration is either impulsive or sustained. The flow is three-dimensional and periodic in the plane normal to the direction of acceleration, and has no external sources. Group theory analysis is applied to accurately account for the mode coupling. Asymptotic nonlinear solutions are found to describe the interface dynamics. The effect of the size of the domain on the diagnostic parameters of the flow is identified. In particular, it is shown that in a finite size the domain the flow is decelerated in comparison to the spatially extended case. The outcomes of the theoretical analysis results for the numerical modeling of the Richtmyer-Meshkov and Rayleigh-Taylor instabilities and for the design of experiments on high energy density plasmas are discussed.

[1]Work is supported by NSF, award 1004330

5:06PM L13.00008 Progress with Incline-Interface Richtmyer-Meshkov Experiments , JACOB MCFARLAND, CHRIS MCDONALD, DAVID REILLY, Texas A&M University, JEFFERY GREENOUGH, Lawrence Livermore National Laboratory, DEVESH RANJAN, Texas A&M University — We describe our progress with a new experiment to investigate Richtmyer-Meshkov instability performed in the newly built variable inclination shock tube at Texas A&M University. In the case of an inclined interface, the amount of vorticity deposited on the initial interface, can be easily controlled by changing the inclination angle, without changing the Mach (pressure gradient) or Atwood number (density gradient). We can achieve this goal by changing the inclination angle of the shock tube. This provides an easy-to-control, clean and repeatable interface for studying the RMI problem. Results will be presented from our initial experiments for a Mach 1.6 shock wave interaction with nitrogen over carbon-dioxide interface for an inclination angle of 60 degrees. Quantitative results such as the interface mixing width growth rate, and vorticity deposition will be discussed in detail. Numerical simulations of the experiments are performed using the ARES code (LLNL) and the time evolution of the interface width, measured from the experiments, is compared to the corresponding numerical predictions.

5:19PM L13.00009 ABSTRACT WITHDRAWN –

Monday, November 19, 2012 3:35PM - 5:45PM –

Session L14 Experiments: PIV II 27B - Chair: David Frakes, Arizona State University

3:35PM L14.00001 PIV Measurement of velocity and acceleration fields using multi-pulse technology[1] , LIUYANG DING, RONALD ADRIAN, Arizona State University, SIVARAM GOGINENI, Spectral Energies, LLC — PIV is extended to the measurement of simultaneous acceleration and velocity fields by the addition of a small number of light pulses. We begin by considering triple-pulse and quadruple-pulse systems, and discuss the optimization of the pulse locations for various measurements. Third-order correlation techniques are applied to the triple-pulse images, yielding significantly improved correlation detectability. Results from experiments in steady flows of know acceleration are used to assess the accuracy and performance.

[1]This work is supported by NAVSEA under STTR contract

3:48PM L14.00002 A Further PIV Validation: The Topological Rule , JOHN FOSS, Michigan State University, DOUGLAS NEAL, LaVision Inc. — Once fully processed, a PIV image provides the velocity vectors (at discrete locations) that are projected onto the plane of the laser light sheet. These discrete vectors are to represent the continuous vector field whose validity can be negated if the vector field does not satisfy the Topological Rule (Foss 2004 and 2007). All, or a portion, of the image can be treated as a collapsed sphere with holes (if appropriate) at the perimeter and handles (if appropriate) in the interior. The Euler characteristic (X) of the selected surface is X = 2 –Sum(holes) – 2Sum(handles) (Eq.1) and X = 2 Sum(nodes) + Sum(1/2 nodes) – 2 Sum(saddles) – Sum(1/2 saddles) (Eq.2) where the "full" singular points are in the interior and the "1/2" singular points are at the perimeter. X from (Eq.1) is known "by construction". X from (Eq.2) is known from an analysis of the resolved vector field of the PIV image. (Note that a given image can be resolved into different regions for different X(1) values.) The Rule is satisfied if X(1) = X(2). Examples of how these concepts have been utilized will be presented. [J. Foss, *Springer Handbook of Experimental Fluid Mechanics*, Chapter C.13, Springer-Verlag, Berlin, 2007.]

4:01PM L14.00003 Multi-planar velocimetry for 3D reconstruction of the flow , AHMAD FALAHATPISHEH, Department of Mechanical and Aerospace Engineering, University of California, Irvine, CA, USA, GIANNI PEDRIZZETTI, Dipartimento Ingegneria Civile e Architettura, Università di Trieste, Italy, ARASH KHERADVAR, Department of Mechanical and Aerospace Engineering, University of California, Irvine, CA, USA — Several extensions of PIV have been proposed for measurements of 3D fields which are restricted for full-volume quantification. We have introduced a fundamentally different solution for experimentally characterizing the incompressible and time-periodic flows in 3D, such as those found in the cardiovascular system. 2D velocity data, acquired by 2C-PIV in multiple planes, is reconstructed to a 3D velocity field taking advantage of the incompressibility of the flow. Using 2D samples instead of scanning the entire 3D domain leads to higher temporal/spatial resolutions since each slice is acquired in a 2D fashion. Hence, there is the possibility of extension to other (medical) imaging modalities that cannot employ advanced 3D optical techniques. 2C-velocimetry on two perpendicular stacks is used for 3D interpolation. The interpolated velocity field is then corrected to satisfy the incompressibility constraint by adding an irrotational velocity field that projects the velocity into a divergence-free vector field space. The method has been validated by exemplary flows having both compact and non-compact structures and different levels of noise. The results show improvements in the reliability of the reconstructed vector field. Application to cardiac flow is also verified.

4:14PM L14.00004 Tomographic PIV using pulsed, high power LED illumination , CHRISTIAN WILLERT, Institute of Propulsion Technology, German Aerospace Center (DLR), 51170 Koeln, NICOLAS BUCHMANN, JULIO SORIA, Department of Mechanical and Aerospace Engineering, Monash University, Melbourne, VIC 3800, Australia — High-power light emitting diode (LED) illumination is investigated as an alternative to traditional laser-based Nd:YAG illumination for Tomographic Particle Image Velocimetry (TPIV). Light pulses of significant intensity (1-10 mJ) are obtained by briefly operating the LED at high drive currents and short pulse durations ($I_f = 24$A, $\tau_p = 150\mu$s). Two LEDs of different wavelengths (green, 525nm; red, 623nm) are investigated with both LEDs providing sufficient pulse energy and image quality to perform reliable TPIV measurements. Volumetric illumination is achieved by direct projection of the LED into the flow, which yields a measurement volume of approximately 8 x 8 x 14 mm^3. The feasibility of this illumination approach is assessed by performing TPIV measurements in homogenous, grid-generated turbulence. in comparison to Nd:YAG laser illumination the absence of laser speckle and excellent pulse-to-pulse stability of the LEDs yield particle image data of high quality with a 3-D displacement measurement uncertainty on the order of 0.2-0.3 pixel. Using an array of LEDs the illuminated volume can be further increased. For instance six high power LEDs are sufficient to illuminate a volume of about 50 x 50 x 10 mm^3.

4:27PM L14.00005 Hands-On Particle Image Velocimetry Experience for Bioengineering Students Using the Interactive Flowcoach System to Understand Aneurysm Hemodynamics , BREIGH N. ROSZELLE, Department of Biological and Health Systems Engineering, Arizona State University and Interactive Flow Studies Corporation, MURAT OKCAY, B. UYGAR OZTEKIN, Interactive Flow Studies Corporation, DAVID H. FRAKES, Department of Biological and Health Systems Engineering and School of Electrical, Computer, and Energy Engineering, Arizona State University — The Flowcoach system is a flow visualization and analysis platform from Interactive Flow Studies that uses particle image velocimetry (PIV) and computational fluid dynamics to provide interactive fluid dynamics education. In the spring of 2012, Flowcoach was used at Arizona State University to help teach bioengineering students about biofluid mechanics. A custom insert was made for Flowcoach to model an anatomical aneurysm that could be treated with a high-porosity flow diverting stent. Students performed PIV on the treated aneurysm model in small lab groups using Flowcoach and then wrote reports comparing their results to those from an untreated aneurysm model. The students were surveyed before and after the project and asked to rate their understanding of general biofluid mechanics, as well as experimental fluid mechanics and aneurysmal hemodynamics. Of the 76 students surveyed, 86% indicated an increase in their understanding of biofluid mechanics, and 90% indicated an increase in their understanding of both PIV and cerebral aneurysm hemodynamics. Students' written feedback showed that they felt Flowcoach and the interactive learning experience it provided were both interesting and beneficial to their future careers as engineers.

4:40PM L14.00006 Development of refractively matched hydrogels for PIV applications , MARGARET BYRON, EVAN VARIANO, University of California, Berkeley — We present a technique for fabricating models whose refractive indices are close to that of water, using two hydrogel polymers. The models' transparency and matched refractive index makes them useful for experiments in Refractive-Index-Matched Particle Image Velocimetry (RIM-PIV). The materials used –polyacrylamide and agarose hydrogel– are inexpensive and can be cast into a variety of desired shapes using injection molding. The models' utility is demonstrated with sets of vector fields, calculated with standard PIV algorithms; vectors can be obtained from the surrounding flowfield and from interior points within the model. Using these data, we calculate solid-body rotation and translation in combination with fluid-phase velocities, and investigate coupling between the two.

4:53PM L14.00007 Devolpment and Evaluation of an Echo Particle Image Velocimetry (EPIV) System[1] , NICHOLAS DEMARCHI, CHRISTOPHER WHITE, University of New Hampshire — An echo particle image velocimetry (EPIV) system capable of acquiring planar fields of velocity in optically opaque fluids or through optically opaque geometries is described, and validation measurements in Hagen-Poiseiulle (pipe) flow are reported. The accuracy limitation and measurement error of the EPIV measurements are assessed by comparing them to theoretically expected flow fields and optical PIV measurements acquired in the same facility. Lastly, the practical issues associated with building a EPIV system are described.

[1]This work is supported by the National Science Foundation, CBET0846359.

5:06PM L14.00008 On the extraction of pressure fields from PIV velocity measurements in turbines , ARTURO VILLEGAS, FANCISCO J. DIEZ, None — In this study, the pressure field for a water turbine is derived from particle image velocimetry (PIV) measurements. Measurements are performed in a recirculating water channel facility. The PIV measurements include calculating the tangential and axial forces applied to the turbine by solving the integral momentum equation around the airfoil. The results are compared with the forces obtained from the Blade Element Momentum theory (BEMT). Forces are calculated by using three different methods. In the first method, the pressure fields are obtained from PIV velocity fields by solving the Poisson equation. The boundary conditions are obtained from the Navier-Stokes momentum equations. In the second method, the pressure at the boundaries is determined by spatial integration of the pressure gradients along the boundaries. In the third method, applicable only to incompressible, inviscid, irrotational, and steady flow, the pressure is calculated using the Bernoulli equation. This approximated pressure is known to be accurate far from the airfoil and outside of the wake for steady flows. Additionally, the pressure is used to solve for the force from the integral momentum equation on the blade. From the three methods proposed to solve for pressure and forces from PIV measurements, the first one, which is solved by using the Poisson equation, provides the best match to the BEM theory calculations.

5:19PM L14.00009 Coherent structure evolution in a turbulent round-jet using scanning tomographic particle image velocimetry , TIERNAN CASEY, King Abdullah University of Science and Technology, Saudi Arabia, JUN SAKAKIBARA, University of Tsukuba, Japan, SIGURDUR THORODDSEN, King Abdullah University of Science and Technology, Saudi Arabia — In order to overcome the inherent spatial resolution limitations and increased noise associated with tomographic PIV when applied to large depth domains, we present a high-speed light-volume scanning technique, using up to 9 adjacent volume slices. This reduces the number of ghost particles while allowing for a large number of total depthwise reconstructed planes, up to 1500. The approach is demonstrated for a turbulent round-jet with Re = 2,500-10,000, using 4 high-speed video cameras to acquire images at up to 1,279 fps, giving over 1 million time-resolved velocity vectors with up to 520 time-steps in sequence. The evolution of tube-like coherent vortical structures are identified and tracked in time across the entire width of the jet - the dynamics of which are compared to existing experimental data using pointwise analysis of velocity gradients.

5:32PM L14.00010 Tomographic PIV measurement in complex geometries of nasal cavity[1] , SUNGHYUK IM, HYUNG JIN SUNG, KAIST, SUNG KYUN KIM, Konkuk University — Flow inside a scaled model of nasal cavity was measured by tomographic PIV. The model was constructed with transparent silicon and a refractive index of working fluid was matched to the model index by mixing glycerol and water. Four cameras and double pulse laser system were used for tomographic PIV. To obtain a high SNR, red fluorescence particles and longpass glass filters were used. Three-dimensional (3D) surface geometry of nasal cavity model from the stereolithography file (.stl) was converted to volume data by adopting the morphological closing and flood-filling algorithm. Coordinates and scaling of the model were adjusted by comparing time series stacking of 3D particle position from volume self-calibration. The geometry information was used to distinguish the fluid and solid region in the tomographic reconstruction procedure. Flow velocity field was calculated from 3D cross-correlation of reconstructed voxel intensities.

[1]This work was supported by the National Research Foundation of Korea(NRF) grant funded by the Korea government(MEST) (No. 2012-0000246).

Monday, November 19, 2012 3:35PM - 5:45PM –

Session L15 Biofluids: Insect Flight 28A - Chair: Tadd Truscott, Brigham Young University

3:35PM L15.00001 Low Reynolds Number Drag Alteration Inspired by Butterfly Scales[1] , BRENT LAFORTE, COURTNEY KRONENBERGER, AMY LANG, University of Alabama — Biomimetics is the process of looking towards nature's adaptations for answers to today's engineering obstacles. An age-old engineering dilemma is trying to find new methods to reduce the amount of drag over a body. This research finds inspiration from butterfly scales which are hypothesized to alter surface friction over the wings. Drop testing was performed on axisymmetric, streamlined, teardrop models which were rapid-prototyped such that the surface was covered with either streamwise or transverse cavities modeled after the Monarch butterfly. The drop tank contained silicone oil with a viscosity two hundred times that of water insuring flow similarity between the model cavities (2.5 mm cavity depth) and the butterfly scale structures (about 30 microns cavity depth). A variation in Reynolds number was achieved by altering the model weight such that terminal speeds ranged from 5 to 70 cm/s. Results showed a reduction in surface friction for the transverse cavity configurations based on the roller-bearing effect. These findings suggest that the cavity shape and ratio is directly correlated to the amount of drag alteration.

[1]Funded by NSF REU grant 1062611.

3:48PM L15.00002 Rock and Roll - How Do Flies Recover From Serial Stumbles? , TSEVI BEATUS, Physics Dept. Cornell University, JOHN GUCKENHEIMER, Mathematics Department, Cornell University, ITAI COHEN, Physics Dept. Cornell University — Flying insects manage to maintain aerodynamic stability despite the facts that flapping flight is inherently unstable and that they are constantly subject to mechanical perturbations, such as gusts of wind. To maintain stability against such perturbations, insects rely on fast and robust flight control mechanisms, which are poorly understood. Here, we *directly* study flight control in the fruit fly *D. melanogaster* by applying mechanical perturbations in mid-air and measuring the insects' correction maneuvers. On each fly we glue a small magnet and use pulses of magnetic field to apply torque perturbations along the fly's roll axis. We then use high-speed filming and 3D reconstruction to characterize the kinematics of their correction maneuver and show how the flies fully recover from roll perturbations of up to 70° within 7-8 wing beats (30-40ms), which is faster than their visual response time. In addition, we study the dynamics of the maneuver by calculating the aerodynamic forces and torques the fly produces. Finally, we present a control mechanism that can explain the roll correction maneuver. These results have implications ranging from the neurobiological mechanisms that underlie flight control to the design of flapping robots.

4:01PM L15.00003 Scales affect performance of Monarch butterfly forewings in autorotational flight[1] , ANYA DEMKO, Reed College, AMY LANG, University of Alabama — Butterfly wings are characterized by rows of scales (approximately 100 microns in length) that create a shingle-like pattern of cavities over the entire surface. It is hypothesized that these cavities influence the airflow around the wing and increase aerodynamic performance. A forewing of the Monarch butterfly (*Danus plexippus*) naturally undergoes autorotational flight in the laminar regime. Autorotational flight is an accurate representation of insect flight because the rotation induces a velocity gradient similar to that found over a flapping wing. Drop test flights of 22 forewings before and after scale removal were recorded with a high-speed camera and flight behavior was quantified. It was found that removing the scales increased the descent speed and decreased the descent factor, a measure of aerodynamic efficacy, suggesting that scales increased the performance of the forewings.

[1]Funded by NSF REU Grant 1062611

4:14PM L15.00004 Flow Modulation and Force Control in Insect Fast Maneuver , CHENGYU LI, HAIBO DONG, WEN ZHANG, KUO GAI, Wright State University — In this work, an integrated study combining high-speed photogrammetry and direct numerical simulation (DNS) is used to study free flying insects in fast maneuver. Quantitative measurement has shown the significant differences between quad-winged flyers such as dragonfly and damselfly and two-winged flyers such as cicada. Comparisons of unsteady 3D vortex formation and associated aerodynamic force production reveal the different mechanisms used by insects in fast turn. This work is supported by NSF CBET-1055949.

4:27PM L15.00005 On the Optimal Dynamic Camber Formation in Insect Flight , YAN REN, HAIBO DONG, Wright State University — It is widely thought that wing flexibility and wing deformation could significantly affect aerodynamic force productions over completely rigid wings in insect flights. However, there is a lack of quantitative discussion of dynamic formation of wing camber and its effect on wing's aerodynamic performance. In this work, a deformable wing is used to model the wing camber and its dynamic formation. A Direct Numerical Simulation (DNS) based computational optimization frame has been developed to obtain the optimal dynamic camber formation of dragonfly in takeoff and cruising flight. Comparative study is then performed between the optimized flexible wing and real dragonfly wing. Results have shown the maximum camber happens around 30% (downstroke) and 80% (upstroke) of one wing beat. Force production and unsteady flows of the flexible wing are also discussed.

4:40PM L15.00006 Quantifying Dragonfly Kinematics During Unsteady Free-Flight Maneuvers[1] , JAMES MELFI, MAE, Cornell University, HUAI-TI LIN, MATTEO MISCHIATI, ANTHONY LEONARDO, HHMI-Janelia Farm, Z. JANE WANG, MAE and Physics, Cornell University — What make dragonflies such interesting fliers are the unsteady high-speed aerial maneuvers they perform. Until recently, the study of dragonflies in mid-flight has been limited to steady-state motions such as hovering and forward flight. In this talk, we report our kinematic analyses of the dragonfly flight recorded in a custom dragonfly arena at HHMI, Janelia Farm. Dragonfly's turning motions often involve all three degrees of freedom about its body axes: yaw, roll, and pitch. We examine the wing kinematics changes associated with different turning maneuvers, and seek the key variables in the wing kinematics that are responsible for each specific maneuver.

[1]This work is supported by a grant to ZJW and AL through the visitor program at Janelia Farm, HHMI.

4:53PM L15.00007 Whole-field, time resolved velocity measurements of flow structures on insect wings during free flight , KENNETH LANGLEY, SCOTT THOMSON, TADD TRUSCOTT, Brigham Young University — The development of micro air vehicles (MAVs) that are propelled using flapping flight necessitates an understanding of the unsteady aerodynamics that enable this mode of flight. Flapping flight has been studied using a variety of methods including computational models, experimentation and observation. Until recently, the observation of natural flyers has been limited to qualitative methods such as smoke-line visualization. Advances in imaging technology have enabled the use of particle image velocimetry (PIV) to gain a quantitative understanding of the unsteady nature of the flight. Previously published PIV studies performed on insects have been limited to velocities in a single plane on tethered insects in a wind tunnel. We present the three-dimensional, time-resolved velocity fields of flight around a butterfly, using an array of high-speed cameras at 1 kHz through a technique known as 3D Synthetic Aperture PIV (SAPIV). These results are useful in understanding the relationship between wing kinematics and the unsteady aerodynamics generated.

5:06PM L15.00008 Mosquito flight failure in heavy fog , ANDREW DICKERSON, Georgia Institute of Technology, LUKE TELLJOHANN, None, LEE-ELLEN THORNTON, CAITLIN MOYER, DAVID HU, Georgia Institute of Technology — Mosquitoes thrive during rainfall and high humidity. We previously found that mosquitoes are successful fliers through rainfall. Heavy fog, consisting of drops three orders of magnitude smaller in mass than raindrops, presents an environment in which mosquitoes cannot maintain flight. Through high-speed videography, we observe mosquitoes reduce wingbeat frequency in heavy fog, but retain the ability to generate sufficient force to lift their bodies, even after significant dew deposition. They are unable, however, to maintain an upright position required for sustainable flight. A mosquito's primary flight control mechanism is its halteres, small knobbed structures evolved from the hind wings, which flap anti-phase with the wings and provide gyroscopic feedback through Coriolis forces. Though the halteres are hydrophobic, repeated collisions with 10-micron fog particles hinders flight control, leading to flight failure.

5:19PM L15.00009 Numerical study of insect free hovering flight , DI WU, KHOON SENG YEO, TEE TAI LIM, National University of Singapore, FLUID LAB, MECHANICAL ENGINEERING, NATIONAL UNIVERSITY OF SINGAPORE TEAM — In this paper we present the computational fluid dynamics study of three-dimensional flow field around a free hovering fruit fly integrated with unsteady FSI analysis and the adaptive flight control system for the first time. The FSI model being specified for fruitfly hovering is achieved by coupling a structural problem based on Newton's second law with a rigorous CFD solver concerning generalized finite difference method. In contrast to the previous hovering flight research, the wing motion employed here is not acquired from experimental data but governed by our proposed control systems. Two types of hovering control strategies i.e. stroke plane adjustment mode and paddling mode are explored, capable of generating the fixed body position and orientation characteristic of hovering flight. Hovering flight associated with multiple wing kinematics and body orientations are shown as well, indicating the means by which fruitfly actually maintains hovering may have considerable freedom and therefore might be influenced by many other factors beyond the physical and aerodynamic requirements. Additionally, both the near- and far-field flow and vortex structure agree well with the results from other researchers, demonstrating the reliability of our current model.

5:32PM L15.00010 Numerical Simulation of *Drophila* Flight Based on Arbitrary Langrangian-Eulerian Method[1] , BELKIS ERZINCANLI, MEHMET SAHIN, Istanbul Technical University — A parallel unstructured finite volume algorithm based on Arbitrary Lagrangian Eulerian (ALE) method has been developed in order to investigate the wake structure around a pair of flapping *Drosophila* wings. The numerical method uses a side-centered arrangement of the primitive variables that does not require any *ad-hoc* modifications in order to enhance pressure coupling. A radial basis function (RBF) interpolation method is also implemented in order to achieve large mesh deformations. For the parallel solution of resulting large-scale algebraic equations, a matrix factorization is introduced similar to that of the projection method for the whole coupled system and two-cycle of BoomerAMG solver is used for the scaled discrete Laplacian provided by the HYPRE library which we access through the PETSc library. The present numerical algorithm is initially validated for the flow past an oscillating circular cylinder in a channel and the flow induced by an oscillating sphere in a cubic cavity. Then the numerical algorithm is applied to the numerical simulation of flow field around a pair of flapping *Drosophila* wing in hover flight. The time variation of the near wake structure is shown along with the aerodynamic loads and particle traces.

[1]The authors acknowledge financial support from Turkish National Scientific and Technical Research Council (TUBITAK) through project number 111M332. The authors would like to thank Michael Dickinson and Michael Elzinga for providing the experimental data.

Monday, November 19, 2012 3:35PM - 5:45PM –

Session L16 Biofluids: Blood Transport 28B - Chair: Jonathan Freund, University of Illinois at Urbana-Champaign

3:35PM L16.00001 The flow of red cells through spleen-like filtering slits , JONATHAN FREUND, University of Illinois at Urbana-Champaign — It is widely understood that the spleen is the principal site in the body for removal of old red blood cells. As they age during their approximately 120 day lifetimes, red blood cells have increasingly slow relaxation times. This mechanical change is potentially the identifying characteristic for filtering in the spleen, which is thought to occur in particularly narrows slit-like passages ($< 1\mu$m $\times \sim 7\mu$m). The mechanism of the filtering, however, is unclear. Most simply, increasing cell viscosity with age would slow, rather than stop, cell passage. Similarly, 'testing' the cells via significant strains during each passage through the spleen might be expected to accelerate aging through fatigue-like mechanisms. Our detailed simulations of red cells passing trough a model slit geometry suggest that increasing cell viscosity can fundamentally change its passage. The results are suggestive of a bifurcation, such as in the onset of instability, with increasing cell interior viscosity. Higher viscosities (or elastic capillary numbers) are seen in cases to lead to a fingering-like instability, which might be expected to severely damage aged cells, leading to their removal, while leaving younger low viscosity cells relatively unstressed.

3:48PM L16.00002 Dynamics of monocytes flowing in a model pulmonary capillary bed[1] , ANNIE VIALLAT, JULES DUPIRE, Adhesion & Inflammation lab / CNRS / Inserm / Aix Marseille University, ADHESION & INFLAMMATION LAB TEAM — The dynamics of blood cells in the pulmonary bed is an issue for tissue perfusion and host defense. The capillary segments in the lungs are smaller than the size of leukocytes so that most of them change their shape to enter and travel through a capillary pathway. During inflammation, changes in the cytoskeleton of leukocytes may stiffen them, resulting in their massive stop and sequestration within lung capillaries. However, due to difficulties of in vivo studies, little is known about the dynamics of leukocytes in the microcirculation and about the coupling between cellular rheology, capillary geometry and flow. We report the dynamics of monocytes (THP-1 cell line) flowing under constant pressure drop in a periodic network of capillaries that mimics the capillary bed. The analysis of cell entrance in the first segment allows the estimation of effective cellular elasticity, viscosity and cortical tension. Cells then present an unsteady regime, with a non-periodic trajectory, a stretching of their average shape and an increase of their velocity. This regime is interpreted from a parameter equivalent to the Deborah number of the system. Finally, a periodic regime is reached with alternatively left and right turns at capillary bifurcations. The reduced cell velocity is governed by an effective friction coefficient between the cell and the capillary walls. Both transient and final regimes depend on cell deformability, as shown by modifying the cortical actin of the cytoskeleton.

[1]This work has been supported by the French Research. National Agency (ANR) under reference ChipCellTrap

4:01PM L16.00003 The effect of polymer additives on flowing cells through capillaries[1] , LAILAI ZHU, KTH Mechanics, Linne flow centre, Royal Institute of technology, Sweden, LUCA BRANDT, KTH Mechanics, Linne flow centre, Royal Institute of technology — It has been suggested that low concentration of long-chain polymer potentially cause benefit in hemodynamics. The effect of polymer additives on blood flow at low Reynolds number is not well studied, albeit well-known for drag reduction in turbulent flow. We adopt a novel general geometry Ewald-like method (GGEM) recently developed[2] to study the fluid-structure interaction. GGEM method can be regarded as an accelerated implementation of boundary integral method, or a variant of immersed boundary method in the Stokesian regime. We perform three dimensional simulations to study the effect of polymer additives on the dynamics of a periodic file of red blood cells (RBCs) through a capillary tube. Fluid motion is solved by spectral element method and solid mechanics of cell membrane by spectral method based on spherical harmonics. Brownian dynamics is used for the polymer molecules.

[1]Funding by VR (the Swedish research council) and Linne flow centre at KTH is acknowledged. Computer time provided by SNIC (Swedish national infrastructure for computing) is also acknowledged.

[2]P. Pranay, *et al.* Physics of Fluids, 22, 123103, 2010

4:14PM L16.00004 Transport of diseased red blood cells in the spleen[1] , ZHANGLI PENG, Department of Materials Science and Engineering, Massachusetts Institute of Technology, IGOR PIVKIN, Institute of Computational Science, University of Lugano, MING DAO, Department of Materials Science and Engineering, Massachusetts Institute of Technology — A major function of the spleen is to remove old and diseased red blood cells (RBCs) with abnormal mechanical properties. We investigated this mechanical filtering mechanism by combining experiments and computational modeling, especially for red blood cells in malaria and sickle cell disease (SCD). First, utilizing a transgenic line for 3D confocal live imaging, in vitro capillary assays and 3D finite element modeling, we extracted the mechanical properties of both the RBC membrane and malaria parasites for different asexual malaria stages. Secondly, using a non-invasive laser interferometric technique, we optically measured the dynamic membrane fluctuations of SCD RBCs. By simulating the membrane fluctuation experiment using the dissipative particle dynamics (DPD) model, we retrieved mechanical properties of SCD RBCs with different shapes. Finally, based on the mechanical properties obtained from these experiments, we simulated the full fluid-structure interaction problem of diseased RBCs passing through endothelial slits in the spleen under different fluid pressure gradients using the DPD model. The effects of the mechanical properties of the lipid bilayer, the cytoskeleton and the parasite on the critical pressure of splenic passage of RBCs were investigated separately.

[1]This work is supported by NIH and Singapore-MIT Alliance for Science and Technology (SMART).

4:27PM L16.00005 Analysis of Red Blood Cell Behavior in a Narrow Tube , HARUKI HOSAKA, TOSHIHIRO OMORI, YOHSUKE IMAI, TAKAMI YAMAGUCHI, TAKUJI ISHIKAWA, Tohoku University — Red Blood Cell (RBC) is a main component of blood accounting for 40 percent in volume, and enclosed by a twodimensional hyper elastic membrane. RBCs strongly influence rheological properties and mass transport of blood. The deformation of RBCs in capillary and at narrowing is also important in considering mechano-transduction of RBCs and hemolysis, though it has not been clarified in detail. Thus, in this study, we investigated the behavior of a RBC flowing in a narrow tube. To carry out the fluid-structure interaction analysis, we coupled a boundary element method to analyze the velocity of the internal and external fluid with a finite element method to analyze the deformation of the membrane. The boundary element method has good calculation accuracy and its computational cost is low because three-dimensional flow filed can be calculated by a two-dimensional computational mesh. The background flow in a tube is pressure-driven Poiseuille flow. Additionally, to reduce the computational time, we implemented massive parallel computation by using GPUs. The results show that the deformation of a RBC is strongly affected by the Capillary number, which is the ratio of viscous force to the elastic force, radius of the tube, and the initial orientation.

4:40PM L16.00006 Mechanistic insights into flow induced segregation in blood and other multicomponent suspensions , AMIT KUMAR, MICHAEL GRAHAM, University of Wisconsin Madison — Blood is a multicomponent suspension comprising mostly of red-blood-cells (RBCs) along with trace amounts of leukocytes and platelets. Under normal flow conditions both the leukocytes and the platelets segregate near the vessel walls, a phenomenon commonly known as margination. The key physical differences between RBCs, leukocytes, and platelets are their relative size and rigidity: leukocytes are larger than RBCs and platelets smaller, but both are considerably stiffer than RBCs. In this work we study the blood flow problem using a model system of fluid-filled elastic capsule mixtures. Using boundary integral (BI) simulations we delineate the effect of size and rigidity on the segregation behavior, and relate these to the observations of leukocyte and platelet margination in blood. Further, we introduce a novel Monte Carlo simulation technique, which incorporates two of the key transport mechanisms in confined suspensions: the wall-induced migration and hydrodynamic pair collisions. The model accurately reproduces the results of BI simulations and provides a mechanistic understanding of the margination phenomena. In particular, it clarifies the important role of heterogeneous pair collisions (collisions between two different species) on the observed margination behavior.

4:53PM L16.00007 ABSTRACT WITHDRAWN –

5:06PM L16.00008 Depletion induced clustering of red blood cells in microchannels , CHRISTIAN WAGNER, MATHIAS BRUST, Experimentalphysik, Saarland University, THOMAS PODGORSKI, GWENNOU COUPIER, Université Grenoble I/CNRS, Laboratoire Interdisciplinaire de Physique — The flow properties of blood are determined by the physical properties of its main constituents, the red blood cells (RBC's). At low shear rates RBC's form aggregates, so called rouleaux. Higher shear rates can break them up and the viscosity of blood shows a shear thinning behavior. The physical origin of the rouleaux formation is not yet fully resolved and there are two competing models available. One predicts that the adhesion is induced by bridging of the plasma (macromolecular) proteins in-between two RBC's. The other is based on the depletion effect and thus predicts the absence of macromolecules in-between the cells of a rouleaux. Recent single cell force measurements by use of an AFM support strongly the depletion model. By varying the concentration of Dextran at different molecular weights we can control the adhesions strength. Measurements at low hematocrit in a microfluidic channel show that the number of size of clusters is determined by the depletion induced adhesion strength.

5:19PM L16.00009 Off-plane motion of a non-spherical capsule in simple shear flow , TOSHIHIRO OMORI, TAKUJI ISHIKAWA, YOHSUKE IMAI, TAKAMI YAMAGUCHI, Tohoku University — Dynamics of a capsule and a biological cell in fluid flow is now of great interest in chemical engineering and bioengineering. In this study, we numerically investigated the motion of a spheroid capsule in simple shear flow including a red blood cell type biconcave disk. The membrane of a capsule was modeled by a two-dimensional hyperelastic material, and its large deformation was solved by a finite element method. The motion of internal and external liquids was estimated as a Stokes flow and solved by a boundary element method. The results showed that the orientation of a spheroid capsule is variant under time reversal, though that of a rigid spheroid is invariant. The final orientation of a spheroid capsule over a long time duration tends to converge to a certain direction depending on the shear rate despite initial placement with random orientation. These results can be utilized for a particle alignment technique and form a fundamental basis of the suspension mechanics of capsules and biological cells.

5:32PM L16.00010 Shear induced diffusion in a red blood cell suspension , THOMAS PODGORSKI, XAVIER GRANDCHAMP, APARNA SRIVASTAV, GWENNOU COUPIER, LIPhy, CNRS-UJF Grenoble — In the microcirculation, blood exhibits an inhomogeneous structure which results in the well know Fahraeus-Lindqvist effect : the apparent viscosity decreases when the diameter of the capillary decreases due to the formation of a marginal cell depletion layer (known as plasma skimming). This structure is a consequence of several phenomena, which include i) the migration of cells aways from walls due to lift forces and gradients of shear and ii) shear induced diffusion due to collisions and interactions among cells. We investigated these phenomena through experiments in simple shear and microchannel flows, with dilute suspensions of vesicles and blood cells. Pairwise interactions between suspended objects result in non-linear and flow-dependent diffusion, whose properties have been measured in different experiments for vesicles and blood cells. The injection of a sheet of concentrated blood cell suspension in a microchannel with a rectangular cross-section allows, through the measurement of its widening along the channel, to measure the diffusivity of blood cells, both in the local plane of shear and in the vorticity direction.

Monday, November 19, 2012 3:35PM - 5:32PM –

Session L17 Biofluids: Microswimmers and Elasticity 28C - Chair: Christophe Eloy, Aix-Marseille University IRPHE

3:35PM L17.00001 Optimal pumping kinematics of a cilium[1] , CHRISTOPHE ELOY, Aix-Marseille University, IRPHE, ERIC LAUGA, Department of Mechanical and Aerospace Engineering, UC San Diego — In a variety of biological processes, eukaryotic cells use cilia to transport flow. Although the internal molecular structure of cilia has been remarkably conserved throughout evolution, experimental observations report qualitatively diverse kinematics in different species. To address this diversity, we have determined numerically the kinematics of the most efficient cilium. Specifically, we have computed the time-periodic deformation of a wall-bound elastic filament leading to transport of a surrounding fluid at minimum energetic cost. Here, the energetic cost is taken to be the sum of positive works done by the internal torques, such that elastic energy is not conservative. The optimal kinematics are found to strongly depend on the cilium bending rigidity through a single dimensionless number, the Sperm number Sp, and closely resemble the two-stroke ciliary beating pattern observed experimentally.

[1]We acknowledge supports from the EU (fellowship PIOF-GA-2009-252542 to C.E.) and the NSF (grant CBET-0746285 to E.L.).

3:48PM L17.00002 Sperm Motility in Flow , JEFFREY GUASTO, GABRIEL JUAREZ, ROMAN STOCKER, MIT — A wide variety of plants and animals reproduce sexually by releasing motile sperm that seek out a conspecific egg, for example in the reproductive tract for mammals or in the water column for externally fertilizing organisms. Sperm are aided in their quest by chemical cues, but must also contend with hydrodynamic forces, resulting from laminar flows in reproductive tracts or turbulence in aquatic habitats. To understand how velocity gradients affect motility, we subjected swimming sperm to a range of highly-controlled straining flows using a cross-flow microfluidic device. The motion of the cell body and flagellum were captured through high-speed video microscopy. The effects of flow on swimming are twofold. For moderate velocity gradients, flow simply advects and reorients cells, quenching their ability to cross streamlines. For high velocity gradients, fluid stresses hinder the internal bending of the flagellum, directly inhibiting motility. The transition between the two regimes is governed by the Sperm number, which compares the external viscous stresses with the internal elastic stresses. Ultimately, unraveling the role of flow in sperm motility will lead to a better understanding of population dynamics among aquatic organisms and infertility problems in humans.

4:01PM L17.00003 Helical bodies swim slower... and faster... through a viscoelastic fluid , SAVERIO SPAGNOLIE, University of Wisconsin-Madison, BIN LIU, THOMAS POWERS, Brown University — Microorganisms frequently swim in fluid environments that exhibit both viscous and elastic qualities in response to deformations. In an effort to better understand the fluid-body interactions in such complex systems, we have studied numerically the force-free swimming of a rotating helix in a viscoelastic (Oldroyd-B) fluid. The introduction of viscoelasticity can either enhance or retard the swimming speed depending on the body geometry and the properties of the fluid (through a dimensionless Deborah number). The results are compared to recent experiments on a rotating helix immersed in a Boger fluid. Our findings bridge the gap between studies showing situationally dependent enhancement or retardation of swimming speed, and may help to clarify phenomena observed in systems ranging from spermatozoan swimming to mechanical drilling.

4:14PM L17.00004 Modeling the swimming of microbes in anisotropic fluids , MADISON KRIEGER, Brown University, SAVERIO SPAGNOLIE, University of Wisconsin, Madison, THOMAS POWERS, Brown University — Microbes commonly swim in non-Newtonian fluids such as mucus, soil, and tissue. Some of these complex fluids are characterized by long-chain molecules which can align, leading to anisotropy. We study a simple model of swimming in an anisotropic fluid, that of an infinitely long two-dimensional sheet deforming via propagating waves and immersed in a nematic liquid crystal. The liquid crystal is categorized by the dimensionless Ericksen number, which compares viscous and elastic effects. At infinite Ericksen number, where viscous effects dominate over elastic effects and the only time scale is the period of the propagating wave, we calculate the swimming speed and power dissipation as a function of the anisotropic viscosities and the tumbling parameter. We also calculate the swimming speed and power dissipation at finite Ericksen number, where the orientation elasticity introduces an additional time scale, the relaxation time.

4:27PM L17.00005 Direct micro-mechanical measurements of the material properties and motility of *C. elegans* , MATILDA BACKHOLM, Department of Physics and Astronomy, McMaster University, Hamilton, ON, L8S 4M1, Canada, WILLIAM S. RYU, Department of Physics, University of Toronto, Toronto, ON, M5S 1A7, Canada, KARI DALNOKI-VERESS, Department of Physics and Astronomy, McMaster University, Hamilton, ON, L8S 4M1, Canada — The model organism *C. elegans*, a millimeter-sized nematode, provides an excellent biophysical system for both static and dynamic mechanical studies. The undulatory motion exhibited by the worm as it swims or crawls through a medium is ubiquitous in nature at scales from microns to meters, and has been the focus of intense research. However, for a successful description of this form of locomotion, a better knowledge of the material properties as well as the worm's output forces is needed. Here we present a new experimental assay, with which the material properties and dynamics of *C. elegans* can be directly probed. In this technique, we use the deflection of a very flexible micropipette to measure the flexural rigidity of *C. elegans* at all stages of its life cycle, as well as along the body of the adult worm. By modelling the worm as a viscoelastic material, we have achieved new insights into its mechanical properties. Furthermore, the forces involved during the undulatory motion of *C. elegans* have been studied. It is the hope that the direct experimental characterization of this model organism will provide guidance for theoretical treatments of undulatory locomotion in general.

4:40PM L17.00006 Effect of rotational diffusion on the collective behavior of swimming microorganisms in viscoelastic fluids , YASER BOZORGI, PATRICK UNDERHILL, Rensselaer Polytechnic Institute — Hydrodynamic interactions of swimming microorganisms can lead to coordinated behaviors of large groups. The impact of viscoelasticity on the collective behavior of active particles driven by hydrodynamic interactions has been quantified with the inclusion of rotational diffusion. Oldroyd-B, Maxwell, and generalized linear viscoelastic modeled are considered as the constitutive equation of the suspending fluid, inspired by some biological fluids. A mean field assumption is used to model the suspension dynamics near an isotropic state. The onset of instability has been quantified by a linear stability analysis in terms of wavenumber, diffusivities, and constitutive equation parameters. Some key results are in contrast to suspensions in Newtonian fluids. The maximal growth rate can occur at a particular wavelength, and diffusion can act to make the system more unstable. Viscoelasticity can also affect the long time dynamics of the continuum equations.

4:53PM L17.00007 Propulsion with a Reciprocal Stroke Enabled by Nonlinear Rheology , PAULO ARRATIA, XIAONING SHEN, NATHAN KEIM, University of Pennsylvania — In a fluid that is entirely viscous, a reciprocal swimming stroke results in no net displacement. However, complex fluids such as mucus or dense suspensions exhibit nonlinear rheology even at low Reynolds number. This nonlinear fluid response can lead to time-reversal symmetry breaking which can enable a reciprocal swimmer to move. Here we demonstrate this principle with a reciprocally-actuated artificial propeller in two viscoelastic fluids: a polymeric fluid with elasticity but negligible shear thinning, and a wormlike micellar fluid that exhibits shear thinning and shear-bands. Propulsion is absent in Newtonian fluid, and is strongest in the shear-thinning micellar fluid. We report on the role of elasticity (Deborah number) in setting the speed of propulsion, and of body shape and boundary conditions in setting its direction. This work is supported by the Army Research Office through award W911NF-11-1-0488.

5:06PM L17.00008 Simulations of micro-swimmer scattering by soft elastic filaments[1] , RODRIGO LEDESMA-AGUILAR, JULIA M. YEOMANS, Rudolf Peierls Centre for Theoretical Physics — The locomotion of microorganisms in the presence of elastic filaments, such as hairs and flagella, is very common in biological systems. We perform a theoretical study, using a simple point-force hydrodynamic model, to analyse the scattering of a dipolar swimmer and semiflexible filaments. Our swimmers consist of active dumbbells that undergo a non-reciprocal swimming stroke leading to locomotion. Fluid-mediated interactions with the elastic chains are modelled using Oseen-level hydrodynamics. We explore the effect of the elasticity of the filaments on the swimmer velocity and orientation.

[1]We acknowledge support from Marie Curie Actions FP7-People-EIF-2010 no. 273406

5:19PM L17.00009 Dynamics of Purcell's three-link microswimmer with a passive elastic tail , EMILIYA PASSOV, YIZHAR OR, Faculty of Mechanical Engineering, Technion, Israel — One of the few possible mechanisms for self-propulsion at low Reynolds number is undulations of an elastic tail, as proposed in the classical work of [Purcell, 1977]. This effect is studied here by investigating a variant of Purcell's three link swimmer model where the front joint angle is periodically actuated while the rear joint is driven by a passive torsional spring. The dynamic equations of motion are formulated and explicit expressions for the leading-order solution are derived by using perturbation expansion. The dependence of the motion on the actuation amplitude and frequency is analyzed, and leading-order expressions are formulated for the travel-per-stroke, mechanical work per travel distance, and average Lighthill's efficiency. Finally, optimization with respect to the actuation frequency and the swimmer's geometry is conducted.

Monday, November 19, 2012 3:35PM - 5:45PM –

Session L18 Biofluids: Respiratory and Aerosols 28D - Chair: Alison Marsden, University of California, San Diego

3:35PM L18.00001 Geometrical influence of pulmonary acinar models on respiratory flows and particle deposition , PHILIPP HOFEMEIER, JOSUE SZNITMAN, Technion - Israel Institute of Technology — Due to experimental challenges in assessing respiratory flows in the deep regions of the lungs, computational simulations are typically sought to quantify inhaled aerosol transport and deposition in the acinus. Most commonly, simulations are performed using generic geometries of alveoli, including spheres, toroids and polyhedra to mimic the acinar region. However, local respiratory flows and ensuing particle trajectories are anticipated to be highly influenced by the specific geometrical structures chosen. To date, geometrical influences have not yet been thoroughly quantified. Knowing beforehand how geometries affect acinar flows and particle transport is critical in translating simulated data to predictions of aerosol deposition in real lungs. Here, we conduct a systematic investigation on a number of generic acinar models. Simulations are conducted for simple alveolated airways featuring a selection of geometries. Deposition patterns and efficiencies are quantified both for massless particles, highlighting details of the local flow, and micron-scale aerosols. This latter group of particles represents an important class of inhaled aerosols known to reach and deposit in the acinus. Our work emphasizes the subtleties of acinar geometry in determining the fate of inhaled aerosols.

3:48PM L18.00002 Direct Numerical Simulation of the Flow in the Human Upper Airway[1] , YONG WANG, SAID ELGHOBASHI, University of California, Irvine — The objective of our study is to understand the flow details in the critical zones inside the human upper airway (HUA) to minimize the guess work in performing surgeries for removing flow obstructions. The 3D flow in HUA consists of unsteady laminar, transitional and turbulent regimes. We perform DNS of HUA flow using lattice Boltzmann method (LBM). We validated our DNS- LBM via comparisons with other DNS methods and experiments for several canonical flows. Excellent agreement was achieved for 3D turbulent channel flow of Kim et al. (JFM 1987) and experimental data for 3D flows in curved pipes. Our predictions of the flow in an idealized HUA model agree well with the experimental data. We predict the flow in a real HUA whose geometry is reconstructed from optical coherence tomography (OCT) data. Both inspiration and expiration cases with various inflow rates are studied. Velocity, pressure and shear stress distributions, time-dependent trajectories of tracer particles and instantaneous streamlines throughout the domain are presented.

[1]This research is supported by NIH grant HL105215-01.

4:01PM L18.00003 Numerical investigation of pulmonary drug delivery under mechanical ventilation conditions[1] , ARINDAM BANERJEE, Lehigh University, Bethlehem, PA, TIMOTHY VAN RHEIN, Missouri University of Science & Technology, Rolla, MO — The effects of mechanical ventilation waveform on fluid flow and particle deposition were studied in a computer model of the human airways. The frequency with which aerosolized drugs are delivered to mechanically ventilated patients demonstrates the importance of understanding the effects of ventilation parameters. This study focuses specifically on the effects of mechanical ventilation waveforms using a computer model of the airways of patient undergoing mechanical ventilation treatment from the endotracheal tube to generation G7. Waveforms were modeled as those commonly used by commercial mechanical ventilators. Turbulence was modeled with LES. User defined particle force models were used to model the drag force with the Cunningham correction factor, the Saffman lift force, and Brownian motion force. The endotracheal tube (ETT) was found to be an important geometric feature, causing a fluid jet towards the right main bronchus, increased turbulence, and a recirculation zone in the right main bronchus. In addition to the enhanced deposition seen at the carinas of the airway bifurcations, enhanced deposition was also seen in the right main bronchus due to impaction and turbulent dispersion resulting from the fluid structures created by the ETT.

[1]Authors acknowledge financial support through University of Missouri Research Board Award.

4:14PM L18.00004 Effect of Pressure Controlled Waveforms on Flow Transport and Gas mixing in a Patient Specific Lung Model during Invasive High Frequency Oscillatory Ventilation , MOHAMMED ALZAHRANY, ARINDAM BANERJEE, Lehigh University, Bethlehem, PA — A computational fluid dynamic study is carried out to investigate gas transport in patient specific human lung models (based on CT scans) during high frequency oscillatory ventilation (HFOV). Different pressure-controlled waveforms and various ventilator frequencies are studied to understand the effect of flow transport and gas mixing during these processes. Three different pressure waveforms are created by solving the equation of motion subjected to constant lung wall compliance and flow resistance. Sinusoidal, exponential and constant waveforms shapes are considered with three different frequencies 6, 10 and 15 Hz and constant tidal volume 50 ml. The velocities are calculated from the obtained flow rate and imposed as inlet flow conditions to represent the mechanical ventilation waveforms. An endotracheal tube ETT is joined to the model to account for the effect of the invasive management device with the peak Reynolds number (Re) for all the cases ranging from 6960 to 24694. All simulations are performed using high order LES turbulent model. The gas transport near the flow reversal will be discussed at different cycle phases for all the cases and a comparison of the secondary flow structures between different cases will be presented.

4:27PM L18.00005 A Comprehensive Breath Plume Model for Disease Transmission via Expiratory Aerosols , S.K. HALLORAN, Dept. Chemical Engineering and Materials Science, Univ. California Davis, A.S. WEXLER, Dept. Mechanical and Aerospace Engineering, Univ. California Davis, W.D. RISTENPART, Dept. Chemical Engineering and Materials Science, Univ. California Davis — The peak in influenza incidence during wintertime represents a longstanding unresolved scientific question. One hypothesis is that the efficacy of airborne transmission via aerosols is increased at low humidity and temperature, conditions that prevail in wintertime. Recent experiments with guinea pigs suggest that transmission is indeed maximized at low humidity and temperature, a finding which has been widely interpreted in terms of airborne influenza virus survivability. This interpretation, however, neglects the effect of the airflow on the transmission probability. Here we provide a comprehensive model for assessing the probability of disease transmission via expiratory aerosols between test animals in laboratory conditions. The spread of aerosols emitted from an infected animal is modeled using dispersion theory for a homogeneous turbulent airflow. The concentration and size distribution of the evaporating droplets in the resulting "Gaussian breath plume" are calculated as functions of downstream position. We demonstrate that the breath plume model is broadly consistent with the guinea pig experiments, without invoking airborne virus survivability. Moreover, the results highlight the need for careful characterization of the airflow in airborne transmission experiments.

4:40PM L18.00006 Correlation among regional ventilation, airway resistance and particle deposition in normal and severe asthmatic lungs[1] , SANGHUN CHOI, ERIC A. HOFFMAN, The University of Iowa, MERRYN H. TAWHAI, The University of Auckland, CHING-LONG LIN, The University of Iowa — Computational fluid dynamic simulations are performed to investigate flow characteristics and quantify particle deposition with normal and severe asthmatic lungs. Continuity and Navier-Stokes equations are solved with unstructured meshes and finite element method; a large eddy simulation model is adopted to capture turbulent and/or transitional flows created in the glottis. The human airway models are reconstructed from CT volumetric images, and the subject-specific boundary condition is imposed to the 3D ending branches with the aid of an image registration technique. As a result, several constricted airways are captured in CT images of severe asthmatic subjects, causing significant pressure drop with high air speed because the constriction of airways creates high flow resistance. The simulated instantaneous velocity fields obtained are then employed to track transport and deposition of 2.5 μm particles. It is found that high flow resistance regions are correlated with high particle-deposition regions. In other words, the constricted airways can induce high airway resistance and subsequently increase particle deposition in the regions. This result may be applied to understand the characteristics of deposition of pharmaceutical aerosols or bacteria.

[1]This work was supported in part by NIH grants R01-HL094315 and S10-RR022421.

4:53PM L18.00007 Numerical simulation of non-equilibrium transient flow during inhalation , OLAF MARXEN, THIERRY MAGIN, von Karman Institute for Fluid Mechanics — The flow in human upper airways may be laminar, transitional, or turbulent. Breadth-by-breadth and patient-specific variability is expected to have a significant influence on laminar-turbulent transition. The flow path of therapeutic drug aerosols may be strongly affected by the transition-induced unsteady structures. The unsteady Navier-Stokes equations are solved numerically to simulate the flow through a channel-flow geometry representative of an airway segment. In order to trigger transition, small-amplitude disturbances are forced via wall blowing/suction. We perform multiple simulations with varying phase of the forced disturbances. Ensemble averaging then allows to compute mean and RMS values. A time-dependent channel center-line velocity serves to model the change in flow velocity during inhalation. The uncertainty associated with variability during breathing is quantified using non-intrusive stochastic collocation. Simulation results reveal that we have intervals in time and space with quasi-steady equilibrium and with strong non-equilibrium flow. The uncertainty associated with the breathing pattern may strongly affect the occurrence of laminar-turbulent transition, leading to large uncertainties when RMS values are peaking.

5:06PM L18.00008 Multiscale Airflow Model and Aerosol Deposition in Healthy and Emphysematous Rat Lungs[1] , JESSICA OAKES, ALISON MARSDEN, Dept. of Mechanical and Aerospace Eng., University of California, San Diego, USA, CELINE GRANDMONT, INRIA Paris-Rocquencourt, France, CHANTAL DARQUENNE, Dept. of Medicine, University of California, San Diego, USA, IRENE VIGNON-CLEMENTEL, INRIA Paris-Rocquencourt, France — The fate of aerosol particles in healthy and emphysematic lungs is needed to determine the toxic or therapeutic effects of inhalable particles. In this study we used a multiscale numerical model that couples a 0D resistance and capacitance model to 3D airways generated from MR images. Airflow simulations were performed using an in-house 3D finite element solver (SimVascular, simtk.org). Seven simulations were performed; 1 healthy, 1 uniform emphysema and 5 different cases of heterogeneous emphysema. In the heterogeneous emphysema cases the disease was confined to a single lobe. As a post processing step, 1 micron diameter particles were tracked in the flow field using Lagrangian particle tracking. The simulation results showed that the inhaled flow distribution was equal for the healthy and uniform emphysema cases. However, in the heterogeneous emphysema cases the delivery of inhaled air was larger in the diseased lobe. Additionally, there was an increase in delivery of aerosol particles to the diseased lobe. This suggests that as the therapeutic particles would reach the diseased areas of the lung, while toxic particles would increasingly harm the lung. The 3D-0D model described here is the first of its kind to be used to study healthy and emphysematic lungs.

[1]NSF Graduate Fellowship (Oakes), Burroughs Wellcome Fund (Marsden, Oakes) 1R21HL087805-02 from NHLBI at NIH, INRIA Team Grant

5:19PM L18.00009 Measurement of ciliary flow generated on the surface of tracheal lumen , KOKI KIYOTA, HIRONORI UENO, TAKUJI ISHIKAWA, KEIKO NUMAYAMA-TSURUTA, YOHSUKE IMAI, TOSHIHIRO OMORI, TAKAMI YAMAGUCHI, Tohoku University — Although we consistently take air with virus and bacteria, these harmful substances are trapped on the surface of tracheal lumen and transported toward larynx from the trachea and bronchi by effective ciliary motion and swallowed it (clearance function). However, the 3-dimensional flow field generated by inhomogeneously distributed ciliary cells are largely unknown. In this study, we first succeeded to measure the ciliated cells' density by staining actin of the epithelial cells and tubulin of the cilia, respectively. Second, we analyzed the ciliary motion by labeling the tip of cilia with fluorescent particles, and tracking their movements to understand the mechanism of the flow generation. Last, in order to clarify the flow field induced by the ciliary motion, we measured the motion of tracer particles on the surface of tracheal epithelial cells by a confocal micro-PTV system. The results show that the mean velocity and the velocity disturbance decayed rapidly as the height from the epithelial cells were increased.

5:32PM L18.00010 Particle Image Velocimetry Measurements in Anatomically-Accurate Models of the Mammalian Nasal Cavity[1] , C. RUMPLE, J. RICHTER, B.A. CRAVEN, M. KRANE, Penn State Applied Research Laboratory — A summary of the research being carried out by our multidisciplinary team to better understand the form and function of the nose in different mammalian species that include humans, carnivores, ungulates, rodents, and marine animals will be presented. The mammalian nose houses a convoluted airway labyrinth, where two hallmark features of mammals occur, endothermy and olfaction. 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 airflow and respiratory and olfactory transport phenomena in anatomically-accurate reconstructions of the nasal cavity. Here, we focus on efforts to manufacture transparent, anatomically-accurate models for stereo particle image velocimetry (SPIV) measurements of nasal airflow. Challenges in the design and manufacture of index-matched anatomical models are addressed and preliminary SPIV measurements are presented. Such measurements will constitute a validation database for concurrent computational fluid dynamics (CFD) simulations of mammalian respiration and olfaction.

[1]Supported by the National Science Foundation.

Monday, November 19, 2012 3:35PM - 5:45PM –

Session L19 Surface Tension III 28E - Chair: Buie Cullen, Massachusetts Institute of Technology

3:35PM L19.00001 Elastocapillary Flows in Flexible Tubes , THERESA HOBERG, MIT, EMILIE VERNEUIL, ESPCI, MIT, ANETTE HOSOI, MIT — Interactions between capillary and elastic effects have sparked interest for a variety of applications, from micro- and nano-scale manufacturing to biological systems. In this work, we investigate capillary flows in extremely flexible, millimeter-scale cylindrical elastic tubes, and examine the interaction between surface tension and elastic effects. We present experimental results for capillary rise and evaporation experiments in different regimes, and demonstrate that surface tension effects on sufficiently flexible tubes can cause them to collapse and coalesce spontaneously through non-axisymmetric buckling. The deformations of the tube wall and their dynamic impact on capillary-driven fluid flow are measured in different regimes, and equilibrium states are characterized. Experiments are compared with theoretical predictions.

3:48PM L19.00002 Sandwiched drops and magnified substrate deformations , JASON S. WEXLER, HOWARD A. STONE, Princeton University — The Laplace pressure and contact line force of a wetting sessile drop (radius R) are strong enough to deform the surface of a soft elastic media (~ 25 kPa) by distances on the order of microns. If the drop, instead of being sessile, is squeezed flat in a gap ($h \ll R$) between two elastic substrates, the resulting forces are much stronger (by a factor $R/h \gg 1$). In fact, a similarly soft material will deform by distances that are orders of magnitude larger than in the sessile drop case. We present an analytical theory that predicts a relationship between drop volume and substrate displacement. In particular, unlike the sessile drop case, where larger deformations correspond to smaller drops, here we find that larger drops lead to greater displacements. We solve for the volume at which the two surfaces first come in contact, and test the predictions of our model with a series of experiments.

4:01PM L19.00003 Capillary forces on elastic solids measured in molecular dynamics , JOOST H. WEIJS, Physics of Fluids, University of Twente, Enschede, ANTONIN MARCHAND, BRUNO ANDREOTTI, Laboratoire de Physique et Mécanique des Milieux Hétérogènes, ESPCI, Paris, JACCO H. SNOEIJER, Physics of Fluids, University of Twente, Enschede — The distribution of capillary forces that a liquid drop exerts on a solid substrate is still debated. While the force normal to the interface can be derived from a global argument, this is not the case for the tangential force component. Experiments in which the force is derived from the elastic deformation of the solid are difficult to perform and interpret, and have lead to different conclusions. To resolve this issue, we directly measure the force in Molecular Dynamics simulations of Lennard-Jones droplets in contact with a solid at varying contact angles. We find that the tangential force component is always pointed towards of interior of the drop, and can qualitatively be explained by density functional theory with the sharp kink approximation. This contradicts the classical view the that the capillary force on the solid acts parallel to the liquid interface.

4:14PM L19.00004 An elastic meniscus , ARNAUD ANTKOWIAK, MARCO RIVETTI, Institut d'Alembert, Université Pierre et Marie Curie and CNRS, Paris, France — A liquid surface touching a solid usually deforms in a near-wall "meniscus" region. In this study, we replace the free surface with a soft polymer and examine the deformation of this "elastic meniscus," result of the interplay between elasticity and hydrostatic pressure. In particular we demonstrate both experimentally and theoretically the existence of a limit height of liquid tenable before collapse of this structure. As a side result, we show that the effort needed to pull an object deposited on a fluid surface is increased with its elasticity, as is common in adhesion phenomena. Finally we discuss the consequences of our results in terms of metrology and optimal tailoring of "elasto-pipettes."

4:27PM L19.00005 Contact angles on a soft solid: from Young's law to Neumann's law , JACCO SNOEIJER, University of Twente, ANTONIN MARCHAND, EPSCI Paris, SIDDHARTHA DAS, University of Alberta, BRUNO ANDREOTTI, EPSCI Paris — The contact angle that a liquid drop makes on a soft substrate does not obey the classical Young's relation, since the solid is deformed elastically by the action of the capillary forces. The finite elasticity of the solid also renders the contact angles different from that predicted by Neumann's law, which applies when the drop is floating on another liquid. Here we derive an elasto-capillary model for contact angles on a soft solid, by coupling a mean-field model for the molecular interactions to elasticity. We demonstrate that the limit of vanishing elastic modulus yields Neumann's law or a slight variation thereof, depending on the force transmission in the solid surface layer. The change in contact angle from the rigid limit (Young) to the soft limit (Neumann) appears when the length scale defined by the ratio of surface tension to elastic modulus γ/E reaches a few molecular sizes.

4:40PM L19.00006 Dynamics of ultrasound-driven two-dimensional microbubbles , CHENG WANG, BHARGAV RALLABANDI, SASCHA HILGENFELDT, Department of Mechanical Science and Engineering, University of Illinois at Urbana-Champaign — Oscillating microbubbles driven by ultrasound are powerful actuators in microfluidics, with applications including mixing enhancement, particle manipulation, and cell lysis. The bubble dynamics is crucial towards the understanding of steady microstreaming flows. We experimentally characterize the oscillation modes and the frequency response spectrum of oscillating bubbles in 2D, driven by a pressure variation resulting from ultrasound in the range of 1 kHz$\leq f \leq$100 kHz. Using high-speed imaging, we analyze the oscillation modes of time-resolved bubble interface shapes. We find that (i) distinct, robust resonance patterns occur independent of details of the set-up, (ii) the position and width of the resonance peaks can be understood using an asymptotic theory approach, and (iii) the appearance of streaming flow patterns is governed by the *relative* amplitudes of bubble surface modes (normalized by the volume response). These results enable an understanding of streaming flow control through tuning of the driving frequency, with consequences for practical applications.

4:53PM L19.00007 Oscillations of a cylindrical bubble attached to a wall , BHARGAV RALLABANDI, CHENG WANG, SASCHA HILGENFELDT, Department of Mechanical Science and Engineering, University of Illinois at Urbana-Champaign — Sustained bubble oscillations in a liquid can result in steady streaming flows that may be exploited as a powerful tool of fluid manipulation at the micron scale. While the oscillations and the resulting streaming of a bubble in bulk fluid are well understood, the practically relevant case of a bubble attached to the wall of a microfluidic device has not been studied extensively, as additional complexity is introduced by the presence of the wall and the bubble contact lines. We provide here an asymptotic theory by which a long-wavelength ultrasound excitation is translated into shape oscillations of the bubble via its surface dynamics. We show that viscous effects near the contact lines entail a coupling mechanism between volume and surface mode oscillations, which governs the features of the frequency response curves obtained. The amplitudes and phases of the bubble oscillation modes are then used to calculate the streaming flow, which is a combined effect of wall-induced streaming and mode-interaction near the surface of the bubble.

5:06PM L19.00008 Straight contact lines on a soft solid , LAURENT LIMAT, Laboratoire Matiere et Systemes Complexes (MSC), UMR 7057 of CNRS and University Paris Diderot — Using a Stokes like approach of incompressible elasticity, I calculated the distortions induced by a straight contact line on a elastic substrate having a non-zero surface tension. The response is very similar to that of Shanahan and de Gennes, except that the short scale divergence of the Log profile is regularized by the elastocapillary length built upon shear modulus and substrate surface tension, and replaced by a Neumann balance of surface tensions. This method is extended to treat a substrate surface tension that differs in the dry and wetted regions, i.e. a contact angle that differs from 90 degrees, with the example of a infinite rivulet composed of two contact lines connected by a curved, cylindrical, liquid interface. The perhaps most surprising result is that the local slope of the substrate, very near contact line, is nearly inversely proportional to the substrate surface tension, each "side" ("dry" or "wet") of the contact line being very close to support one half of the applied vertical force. This result has surprising implications for the approximate treatment of wetting hysteresis on substrates having both plastic and elastic properties, that seem to be ruled at small scales only by surface tension effects

5:19PM L19.00009 Multiple equilibria and evaporation in elastocapillary systems , KIRAN SINGH, MICHELE TARONI, DOMINIC VELLA, OCCAM, Oxford, TAE-HONG KIM, HO-YOUNG KIM, Seoul National University, Korea — In this talk we consider the coalescence of microscopic flexible beams caused by the surface tension of an intervening liquid. Our work is motivated by the coalescence observed in the manufacture of MEMS devices. We develop a model coupling the elastic deflection of the beams, lubrication-type flow and surface tension; we also investigate the role played by the evaporation of the liquid. In the absence of evaporation, multiple equilibrium states exist. We characterize these states and determine their stability. We consider the effect of evaporation and show that sufficiently high evaporation rates can suppress the tendency of beams to stick together. Finally, we consider some extensions of our models showing how the interaction between multiple beams may delay the rate of evolution and discuss the way in which heating the beams is even more effective in suppressing sticking.

5:32PM L19.00010 Capillary rise of oil in an aqueous foam , KEYVAN PIROIRD, ÉLISE LORENCEAU, Laboratoire Navier, UMR 8205 du CNRS, Champs-sur-Marne, France — Oil is usually known as an anti-foaming agent. Yet, it has been shown that oil droplets present in the foaming solution can have the opposite effect and stabilize a foam when unable to cross the air/water interface. In these previous studies, oil is first emulsified and then mixed with air to generate a foam. In this work, we report experiments where an aqueous foam is put in direct contact with a large oil drop. With the appropriate choice of oil and surfactants, oil spontaneously invades the liquid network of the foam without damaging it. We study the dynamics of penetration at the scale of a single Plateau border, that acts as a "liquid capillary tube" in which oil flows in an unbroken stream. At the end of the experiment, a long and stable cylinder of oil is formed in the Plateau border. This cylinder breaks up into droplets when, following a rearrangement, oil is transferred from the Plateau border to a soap film.

Monday, November 19, 2012 3:35PM - 5:32PM –

Session L20 Turbulent Boundary Layers VII: Rough Walls I 30A - Chair: Kenneth Christensen, University of Illinois at Urbana-Champaign

3:35PM L20.00001 Structural Aspects of Flow Over Highly Irregular Roughness Revealed from Wall-Normal–Spanwise Plane Stereo PIV Measurements , J.M. BARROS, K.T. CHRISTENSEN, University of Illinois at Urbana-Champaign — The structural attributes of turbulent flow over a complex roughness topography were explored with both low-frame-rate and time-resolved stereo particle-image velocimetry in a wall-normal–spanwise ($y-z$) measurement 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. Instantaneous velocity fields in the cross-flow measurement plane revealed structural attributes qualitatively consistent with smooth-wall flow structure, particularly patterns of spanwise-alternating, large-scale regions of low and high streamwise momentum. However, single-point turbulence statistics revealed significant statistical heterogeneity in the form of low- and high-momentum flow pathways marked by enhanced Reynolds stresses and turbulent kinetic energy. The low-momentum flow pathways were also marked by intense vortical activity along their spanwise boundaries, indicating that these pathways could represent preferential "channeling" of large-scale motions due to the roughness below or the generation of "trains" of vortical structures shed from the roughness that advect along a common path downstream.

3:48PM L20.00002 Roughness effects in turbulent channel flow for the transitionally rough regime[1] , KAREN FLACK, MICHAEL SCHULTZ, United States Naval Academy — Skin friction coefficients and roughness functions are presented for sandpaper roughness (220, 320 and 500 grit) throughout the transitionally rough regime from hydraulically smooth to fully rough. The experiments were performed in a 8:1 aspect ratio water channel with a Reynolds number based on channel height range of 12,000 - 280,000. The wall shear stress was determined from the pressure drop in the channel. Mean flow and turbulence statistics were also obtained for the 320 grit sandpaper using a two-component LDV at Karman numbers ranging from Re_τ=1000 - 6000. The mean flow and turbulence data are compared to smooth wall channel results from experiments and DNS at similar Re_τ.

[1]Work supported by the Office of Naval Research

4:01PM L20.00003 Effects of increased entrainment in turbulent boundary layers[1] , GUILLEM BORRELL, JAVIER JIMENEZ, Universidad Politecnica de Madrid — It has been reported that certain rough surfaces modify the outer region of turbulent boundary layers. One of the effects of surface roughness is additional friction that causes an accelerated entrainment rate, which is also known to modify the outer intermittent layers of external turbulent flows. One and two-points statistics are presented from a direct numerical simulation of a zero-pressure-gradient turbulent boundary layer in the range $Re_\theta = 1400 - 4500$, in which the spreading rate is increased by 70% by a smooth volumetric force restricted to the layer below $y^+ = 25$, and equivalent to a sand roughness of $k_s \sim 60$. The goal of this simulation is to separate the effects of surface geometry from those of entrainment. The velocity fluctuations, Reynolds stresses and spatial correlations $C_{\xi\xi}(x;x',y;y',k_z)$, that are consistently different from those in smooth-wall boundary layers at similar Reynolds numbers, will be compared with experimental and numerical data sets available in the literature.

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

4:14PM L20.00004 Time Resolved Tomographic PIV Measurements of Rough-Wall Turbulent Channel Flow[1] , RINALDO MIORINI, CAO ZHANG, PATRICK LUCKETT, DEV PATEL, JOSEPH KATZ, Johns Hopkins University — Turbulent channel flow is investigated via time-resolved tomographic particle image velocimetry. The optical refractive index of the transparent channel rough wall is matched with that of the fluid, allowing measurements very close to its surface. A thick, high-speed laser sheet illuminates tracers whose scattered light is recorded by four high-speed cameras. The roughness consists of staggered pyramidal elements whose aspect ratio satisfies the "well-characterized" flow conditions, with h/k close to 50 and k+=60-100 (h and k are the channel half-height and roughness height, respectively). The measurements are performed at Re(h) in the 40000-60000 range. Following the work of Hong et al. (JFM, 2011, 2012), data analysis is aimed at understanding the interaction between outer layer large-scale structures and the roughness elements. We examine the processes involved with vortex generation at the pyramid front ridge, vortex evolution in the non-uniform flow in the roughness sub-layer, its rise as neighboring structures interact, and its subsequent development under the influence of outer layer structures. Associated trends of Reynolds stresses and TKE are also explored, taking advantage of the available three-dimensional velocity gradients.

[1]Sponsored by NSF

4:27PM L20.00005 Large-Scale Secondary Flows in a Turbulent Boundary Layer Caused by Highly Ordered and Directional Surface Roughness , BAGUS NUGROHO, NICK HUTCHINS, JASON MONTY, The University of Melbourne — The effects of highly-ordered and directional surface roughness on zero-pressure-gradient turbulent boundary layers has been investigated experimentally. The surface roughness geometry is a converging-diverging riblet-type, which is shown to induce large-scale secondary flows within the boundary layer. Detailed studies using a single-normal hot-wire indicate that the surface roughness promotes large-scale spanwise modifications of the boundary layer characteristics on the wavelength of the converging-diverging pattern. The data reveals that the local mean velocity above the diverging region increases, while the turbulent intensity and boundary layer thickness decreases. The opposite phenomena occurs over the converging region. Parametric studies reveal that the magnitude of the induced spanwise modifications is determined by the viscous scaled riblet height and spacing (h^+ and s^+), the converging / diverging angle (α), and the streamwise fetch of the boundary layer flow over the surface roughness.

4:40PM L20.00006 Investigation of wall-bounded turbulence over regularly distributed roughness , MARCO PLACIDI, BHARATHRAM GANAPATHISUBRAMANI, University of Southampton — The effects of regularly distributed roughness elements on the structure of a turbulent boundary layer are examined by performing a series of Planar (high resolution $l^+ \approx 30$) and Stereoscopic Particle Image Velocimetry (PIV) experiments in a wind tunnel. An adequate description of how to best characterise a rough wall, especially one where the density of roughness elements is sparse, is yet to be developed. In this study, rough surfaces consisting of regularly and uniformly distributed LEGO® blocks are used. Twelve different patterns are adopted in order to systematically examine the effects of frontal solidity (λ_f, frontal area of the roughness elements per unit wall-parallel area) and plan solidity (λ_p, plan area of roughness elements per unit wall-parallel area), on the turbulence structure. The Karman number, Re_τ, is approximately 4000 across the different cases. Spanwise 3D vector fields at two different wall-normal locations (top of the canopy and within the log-region) are also compared to examine the spanwise homogeneity of the flow across different surfaces. In the talk, a detailed analysis of mean and rms velocity profiles, Reynolds stresses, and quadrant decomposition for the different patterns will be presented.

4:53PM L20.00007 Drag and Turbulence Production by Random Roughness , RICHARD LEIGHTON, None, KENNETH CHRISTENSEN, University of Illinois, Urbana-Champaign, KIRAN BHAGANAGAR, University of Texas, San Antonio — The effects of roughness in an incompressible turbulent boundary layer include the increased production of turbulence kinetic energy (TKE) and altered the nature and distribution of the skin drag. By formulating the exact Reynolds-averaged Navier-Stokes turbulence kinetic energy equations in a manner that includes an arbitrary roughness, the averaged terms representing the roughness production of TKE and the roughness drag can be written explicitly. Similar transport equations for TKE can be formulated wherein the roughness geometry is represented using the immersed boundary methodology. These terms are calculated from a collection of direct numerical simulations (DNS). The roughness geometry employed is based on both real turbine blade roughness and a set of spectrally defined random roughness with varying amounts of skew. The primary results include an examination of thr distribution of the roughness drag and the partitioning of the production of TKE into canonical shear production and into production by roughness, and the partitioning of drag into form drag and viscous shear drag.

5:06PM L20.00008 Holographic particle tracking elucidates coherent structures in the roughness sublayer of a channel flow[1] , SIDDHARTH TALAPATRA, JOSEPH KATZ, Johns Hopkins university — The 3D, volumetric flow in the inner part of a turbulent rough channel, at Re_τ=3520, is resolved using microscopic holographic particle tracking in an optically index-matched facility. The channel walls consist of uniformly distributed pyramids, with height k=0.46mm, k^+=65 and h/k=54 (h is the channel half height). Mean velocity and Reynolds stress profiles agree with 2D PIV results except for very close to the wall ($< 0.7k$), where discrepancies are attributed to the higher resolution of the holographic data. All the Reynolds stress components increase rapidly within the roughness sublayer as the wall is approached. Instantaneous realizations indicate that the roughness sublayer is flooded by low lying spanwise, groove parallel and quasi-streamwise vortices. Linear stochastic estimation and conditional sampling reveal that the prevalent sublayer structure consists of interacting U-shaped vortices with spanwise base located in the low speed region above the pyramid ridgeline, and quasi streamwise legs extending between ridgelines, where the velocity is higher. Interactions among legs of vortices generated above neighboring pyramids induces ejection, lifting the quasi-streamwise legs and aligning them preferentially at angles of 54°-63° to the streamwise direction.

[1]Sponsored by ONR and NSF

5:19PM L20.00009 Extraction of very-large scale structures in turbulent boundary layer , STÉPHANE ROUX, Université de Nantes, FRANCK KERHERVÉ, MICHEL STANISLAS, JEAN MARC FOUCAUT, Ecole Centrale de Lille, JOEL DELVILLE, CNRS PPRIME, LML ER2 TEAM, PPRIME ATAC TEAM — The examined flow is a zero-pressure gradient turbulent boundary layer. The data used are taken from the joined experimental campaign conducted during the european WALLTURB program in the large wind tunnel at Laboratoire de Mécanique de Lille (LML). The free-stream velocity is 10 m/s. At the investigated position, the boundary layer thickness is 30 cm and the Reynolds number based on the momentum thickness is 19100. A methodology for eduction of *super-structures* is presented. These structures are characterised by a large degree of persistance and are thought to participate actively to the turbulence regeneration in the near-wall region (Marusic et al. 2010. A time-resolved estimate of the three-dimensionnal structures is obtained by combining low-speed two-dimensional stereo-PIV at 4Hz and a two-dimensionnal rake of 143 single hot-wire probes at 30 kHz. The very large scale structures are clearly reconstructed which exhibit a streamwise extent an order of magnitude larger than the boundary layer thickness. Interest is particulary focused on the low-speed species of these structures. Associated coounter-rotating vortices are also evidenced in good agreement with the litterature.

Monday, November 19, 2012 3:35PM - 4:53PM –

Session L21 Turbulent Boundary Layers VIII: Rough Walls II 30B - Chair: Beverley McKeon, California Institute of Technology

3:35PM L21.00001 Roughness effects on the control of laminar separated boundary layers[1] , AYSE GUL GUNGOR, Istanbul Technical University Faculty of Aeronautics and Astronautics, MARK PHIL SIMENS, School of Aeronautics, Universidad Politécnica de Madrid — The effect of roughness positioned upstream of a separation bubble is studied by direct numerical simulation using a high resolution numerical scheme, and the immersed boundary method. The two-dimensional roughness elements have rectangular shapes, and a height h between 0.9 and $1.8\theta_0$, θ_0 being the inflow momentum thickness. The preliminary 2D studies show a decrease in the extent of the separation bubble as well as the separation position compared with the baseline case without surface roughness. The 3D studies show an earlier transition which eliminates the laminar separation bubble completely. Furthermore, it shows that downstream of the position where the original separation bubble reattaches, the skin friction, C_f follows the same functional behavior as the C_f in the uncontrolled flow, although having a higher value. Further work will be done in trying to optimize the roughness based control, which consists in minimizing the profile losses by finding a balance between the reduction of the extent of the separation bubble and the transition to turbulence.

[1]Funded by ITU

3:48PM L21.00002 Direct numerical simulation of turbulent boundary layer separation under unsteady pressure gradients[1] , WILLIAM BROMBY, DONGHYUN YOU, Department of Mechanical Engineering, Carnegie Mellon University — Direct numerical simulations of attached, separated, and unsteady separated turbulent boundary layers are performed. Blowing-suction velocity distributions are imposed along the upper boundary to introduce adverse pressure gradients to the turbulent boundary layer. A time varying adverse pressure gradient induces unsteady separation of the turbulent boundary layer. Comparing unsteady and steady separated cases demonstrates significant differences in distributions of the average velocity, vorticity, and kinetic energy budget despite similar average wall pressure and skin friction distributions. The behavior of the unsteady separated turbulent boundary layer is described using instantaneous flow visualization, frequency of flow reversal, turbulent kinetic energy budget, vorticity distributions, Reynolds stress events, and auto-correlation of velocity fluctuation in time and space. Complex flow phenomena such as division of the recirculation zone during separation bubble collapse, motion of vortical structures into and over the bubble, and characteristics of detachment point and reattachment point variation in stream and span are revealed.

[1]Supported by the Army Research Office Grant W911NF1010348

4:01PM L21.00003 Turbulent flow around a wall mounted square cylinder: evaluating the LES and Reynolds Stress turbulence models in predicting the negative turbulence productions upstream the obstacle , BEHTASH TAVAKOLI, GOODARZ AHMADI, Department of Mechanical and Aeronautical Engineering Clarkson University, Potsdam NY 13699 — The airflow field around a square cylinder was simulated using the Reynolds Stress Turbulence Model (RSTM) as well as the Large Eddy Simulation (LES). Particular attention was given to the case with Reynolds number of 5610 for which the Direct Numerical Simulation (DNS) data of Yakhot et al (2006) were available. They found that the unsteadiness due to the unstable interaction of the flow at the upstream and the sides of the cube generated the vortex shedding downstream the cube. Also, they argued that the inaccuracy of some of the RANS and LES turbulence models was because of their inability to predict the negative turbulence production in front of the cube, which was the source of the horseshoe vortex. Therefore, in this paper the accuracy of the RSTM and LES turbulence models in predicting the turbulence production at the upstream of the wall mounted square cylinder was investigated. The nature of the 3D wakes behind the cube as well as the vortices in front and at the back of the cube were analyzed. The simulation results were compared with the DNS data and the accuracy of the two turbulence models were underlined.

4:14PM L21.00004 Large eddy simulation of channel flows with strip roughness[1] , NAMIKO SAITO, DALE PULLIN, California Institute of Technology — We describe a large-eddy-simulation (LES) study, at Reynolds number up to $Re_\tau = O(10^8)$, of turbulent flow in a long channel with the walls consisting of roughness strips. The strips are oriented either parallel or perpendicular to the flow resulting in repeated transitions of smooth and rough surfaces in the span-wise or stream-wise directions respectively. The present LES uses a wall model which contains Colebrook's empirical formula as a roughness correction to both the local, dynamic calculation of u_τ and also the LES wall boundary condition. This operates point-wise across wall surfaces, and hence changes in the outer flow can be viewed as a response to the overall roughness distribution. The results indicate that for the strips perpendicular to the flow, the recovery of the flow beyond a smooth to rough transition is faster than that of a rough to smooth transition.

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

4:27PM L21.00005 Direct numerical simulations in turbulent boundary layers over cube-roughened walls with varying spanwise spacing[1] , JUNSUN AHN, JAE HWA LEE, HYUNG JIN SUNG, KAIST — Direct numerical simulations of turbulent boundary layers over three-dimensional cube-roughened walls were performed to investigate the effects of the spanwise spacing (p_z/k) on the turbulent statistics. The spanwise extent between the cubes was varied p_z/k=2, 3, 4 and 6 at the fixed streamwise extent (p_x/k=3), where k is the roughness height. Lee *et al.* (2012) examined by varying the streamwise extent (2$\leq p_x/k \leq$10) that the roughness function has the maximum contribution at p_x/k=4 and the outer value of the Reynolds stresses at $y/\delta \approx$0.4 increases with increasing the streamwise pitch (p_x/k). The roughness function has the local maximum at p_z/k=3 and the value of the Reynolds stresses at the same outer location increases linearly with increasing the spanwise pitch (p_z/k). This implies that the wall friction is closely correlated with the roughness density because the roughness function has the maximum at the similar roughness density and there is also a strong interaction between the inner and outer regions at large spacing values of p_z/k. Furthermore, we can understand the roughness effects on the turbulence structures.

[1]This work was supported by the Creative Research Initiatives program (No. 2012-0000246) of National Research Foundation and was partially supported by KISTI under the Strategic Supercomputing Support Program

4:40PM L21.00006 Large Eddy Simulation of fully developed turbulent flow over a moving wavy surface , LIMING SUN, SHUYANG CAO, SKLDR, Tongji University, China — In order to investigate the vertical profile of wind speed over waves during typhoons for wind engineering applications, Large eddy simulation is performed to analyze the fully developed turbulent flow over a moving wavy surface. The Reynolds number Re is 10170, where Re is based on the channel depth δ and bulk velocity U. The wave steepness $2\pi\alpha/\lambda$ (α and λ is wave amplitude and wavelength respectively) is 0.25. Different wave age c/U that ranges from -1.0 to 2.0 is considered to model the influence of c/U on flow separation, drag force and vertical profiles of mean velocity and turbulence intensity. The simulation considered two kinds of flow motions, translation and undulation, in other words, whether the rigid wavy surface translates downstream with velocity c ($>$0), or the wary surface moves downstream as a deformable undulation with phase speed c. Comparisons of the mean and turbulent velocity characteristics between these two flow motions are made to illustrate the difference of turbulence generation mechanism of these two motions, while shedding lights on the clarification of wind profile for wind engineering applications.

Monday, November 19, 2012 3:35PM - 5:06PM –

Session L22 Turbulence Mixing III 30C - Chair: Katepalli Sreenivasan, New York University

3:35PM L22.00001 Turbulent mixing of substances which are highly diffusive[1] , K.R. SREENIVASAN, New York Univ., P.K. YEUNG, Georgia Tech — How a substance gets mixed by a fluid, even when the motion is turbulent, depends to some extent on whether its diffusivity is small or large. The magnitude of the diffusivity is usually expressed by the Schmidt number (Sc, ratio of fluid viscosity to the diffusivity of the substance). The case of passive scalars (which have no back-reaction on the flow) with large Sc (weak diffusivity) has received considerable attention, especially for its special features such as the -1 power roll-off of the spectrum of the fluctuations. Similar studies for passive scalars at low Schmidt numbers (or large diffusivity) do not yet exist, though the classical theory (Batchelor, Howells & Townsend, J. Fluid Mech, **5**, 134 (1959)) is now more than fifty years old. In this talk we report direct numerical simulations for decaying scalar fields with Sc as low as 1/2048, at grid resolution up to 4096^3, in stationary isotropic turbulence with microscale Reynolds number in the range 140-390. We examine the validity of theoretical assumptions that lead to a spectral slope of -17/3 in the so-called inertial-diffusive range. Despite limitations on the range of scales in the simulations, the data support the theory as the Schmidt number decreases and the Reynolds number increases.

[1]Supported by NSF Grant CBET-1139037.

3:48PM L22.00002 Turbulent mixing of highly diffusive substances in the presence of a uniform mean gradient[1] , P.K. YEUNG, K.P. IYER, Georgia Tech, K.R. SREENIVASAN, New York Univ. — In the preceding talk, we verified the basic physical content and general validity of the theory of Batchelor, Howells & Townsend (J. Fluid Mech. **5**, 134 (1959)) in the case of passive scalar fields decaying in the absence of any production mechanism. In this study we consider results from direct numerical simulations on passive scalars of very strong diffusivity (low Schmidt number), in a presence of a uniform mean gradient, which leads to production of scalar fluctuations at the large scales. Our results show that the presence of the mean gradient alters the physics of mixing fundamentally. While the spectrum of scalar fluctuations still follows a -17/3 power law in the so-called diffusive-inertial range, the constant is found to be non-universal and dependent on the magnitude of the mean scalar gradient. Spectral transfer activity is greatly reduced compared with that for moderately and weakly diffusive scalars. The resulting weakening of spectral cascade at high wavenumber leads to many distinctive features, including the failure of dissipative anomaly, and a new balance of terms in the spectral transfer equation for the scalar variance, different from the case of no gradient. We also present an alternative explanation for the scaling behaviors actually observed.

[1]Supported by NSF Grant CBET-1139037.

4:01PM L22.00003 Turbulent mixing in microfluidics with Reynolds number in the order of 1 , FANG YANG, WEI ZHAO, GUIREN WANG, University of South Carolina — One important issue in microfluidic devices is the relatively slow mixing due to laminar flow at low Reynolds number (Re). In many cases, fast mixing is highly demanded. In macroscale, where Re is relatively high, mixing can be enhanced by forcing flow to be turbulent. However, although there can be elastic turbulence in low Re, it is conventionally believed that the flow in mirofluidics, where typical Re is in the order of 1 or less, can only be laminar. In present work, we demonstrate that turbulent mixing can be realized in a microchannel, where the Reynolds number is in the order of 1 when the flow is forced electrokinetically. The turbulent mixing in microfluidics can cause ultrafast scalar mixing. Confocal microscopic laser induced fluorescence with high tempo-spatial resolution is used to study the turbulent mixing in the microchannel. We report the fast mixing process, concentration profile, irregular concentration time trace, segregation intensity and continuous power spectrum of concentration fluctuation indicating multiscale structures of small eddies. The results indicate that turbulent mixing can be realized as well in microfluidics with Re in the order of 1. The study could open a new perspective view on transport phenomena in microfluidics.

4:14PM L22.00004 Spinodal turbulence enhances heat transfer in micro devices , STEFANO FARISÉ, PIETRO POESIO, GIAN PAOLO BERETTA, Università degli Studi di Brescia — We experimentally prove the possibility of using spinodal mixtures to increase heat transfer in micro devices as a consequence of an evenly distributed micro agitation, which increases the effective diffusivity. Despite the Re-number is as low as 5, turbulence-like mixing can be achieved by mass transfer effects. A mixture of acetone-hexadecane is quenched in a micro heat exchanger to induce spinodal decomposition. The heat transfer rate is enhanced by self-induced convective motion (spinodal turbulence) because the drops of one phase move against each others under the influence of non-equilibrium capillary forces, Korteweg stresses,which are sustained by the free energy liberated during phase separation. The heat transfer is increased up to the 200% and the effect become larger as the bulk Re decreses, while no dramatic increase in the pressure drop is observed. We built two different experimental set-ups: in the first we measure the heat transfer with a feedback method and in the second we measure the pressure drop and we visualize the induced convection. High-speed camera visualization,pressure drop and temperature measurements allow a complete characterization of the phenomenon, with a special attention to the quantification of the heat transfer coefficent enhancement.

4:27PM L22.00005 Spectral transfer and scale locality characteristics in turbulent mixing over a wide range of Schmidt numbers[1] , DHAWAL BUARIA, P.K. YEUNG, Georgia Tech, J.A. DOMARADZKI, Univ of Southern Calif. — A classical picture of turbulent mixing is that advective transport by the velocity field causes blobs of scalar fluctuations to be broken down into smaller and smaller scales, where the fluctuations are ultimately dissipated by molecular diffusivity. In Fourier space this corresponds to a spectral cascade, which is generally understood to be local in nature (i.e., occurring among scales similar in size). However, recent numerical simulations show that at very low values of the Schmidt number (Sc) the spectral cascade is strongly suppressed. To understand this observation we examine the detailed spectral transfer characteristics of scalar fields with Sc ranging from 1/2048 to 64 in isotropic turbulence with Taylor-scale Reynolds number 140. We also compute so-called scale locality functions which measure contributions from "resolved" and "subgrid" scales to the transfer flux across a specified cutoff scale. Our results suggest that the classical cascade scenario holds well at $Sc \geq 1$. However, in the limit $Sc \ll 1$ transfer is dominated by nonlocal triadic interactions involving low wavenumber scalar modes and high wavenumber scalar modes, modulated by a high wavenumber velocity mode, corresponding to advection by the small-scale velocity field in physical space.

[1]Supported by NSF Grant CBET-1139037.

4:40PM L22.00006 Extreme responses of a coupled scalar-particle system during turbulent mixing[1] , JOERG SCHUMACHER, BIPIN KUMAR, Ilmenau University of Technology, Ilmenau, Germany, RAYMOND SHAW, Michigan Technological University, Houghton, USA — Extreme responses of a droplet ensemble during an entrainment and mixing process as present at the edge of a cloud are investigated by means of three-dimensional direct numerical simulations. We combine therefore the Eulerian description of the turbulent velocity and vapor content fields with a Lagrangian ensemble of cloud water droplets which are advected in the flow and can shrink and grow in correspondence with the supersaturation at their position. We find that the Damköhler number Da, a dimensionless parameter which relates the fluid time scale to the typical evaporation time scale, can capture all aspects of the initial mixing process. The mixing process is characterized by the limits of strongly homogeneous ($Da \ll 1$) and strongly inhomogeneous ($Da \gg 1$). We explore these two extreme regimes and study the response of the droplet size distribution to the corresponding parameter settings through enhancement and reduction the response constant K in the droplet growth equation. Thus, Da is varied while Reynolds and Schmidt numbers are held fixed, and initial microphysical properties are held constant. In the homogeneous limit minimal broadening of the size distribution is observed as the new steady state is reach, whereas in the inhomogeneous limit the size distr

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

4:53PM L22.00007 Lyapunov exponents of inertial particles in isotropic turbulence , LI GUO, GUODONG JIN, DONG LI, GUOWEI HE, Chinese Academy of Science, TURBULENCE TEAM — The Lyapunov exponents of inertial particles in isotropic turbulence are calculated using direct numerical simulation (DNS), filtered DNS and large-eddy simulation (LES). Here, filter operators are taken as Eulerian space filter and Lagrangian time filter. The Lyapunov exponents obtained are qualitatively consistent but their magnitudes are different: the Lyapunov exponents from DNS are largest and the ones from LES are smallest while the ones from filter DNS are in between. The comparisons imply that the filters could reduce both stretching and compression in turbulent flows. Furthermore, the Lagrangian time filter allows the filtered trajectories to share the similar statistics of true particle trajectories in turbulent flows.

Monday, November 19, 2012 3:35PM - 5:32PM –

Session L23 Turbulent Boundary Layers IX: Rough / Wavy / Bent Walls 30D - Chair: Jonathan Naughton, University of Wyoming

3:35PM L23.00001 Index-Matched PIV Measurements of Turbulence inside a Fractal-Tree Canopy[1]

KUNLUN BAI, JOSEPH KATZ, CHARLES MENEVEAU, Johns Hopkins University — Turbulence inside vegetation canopies has a significant impact on various physical and biological processes such as forest microclimate, rainfall evaporation distribution and climate change. In most scaled laboratory experimental studies, the canopy element models, for example vertical strips or rods, typically have only one or a few characteristic length scales. However, natural canopies usually contain multiple scales with branches and sub-branches. In this study, a model canopy is constructed by twelve fractal-like trees. Each tree contains five generations with three branches and a scale reduction factor 1/2 at each generation and fractal similarity dimension of Df $\sim$ 1.58. In order to capture the flow fields inside the trees and between the branches, an index-matching technique is applied. Two trees are made by clear urethane plastic with refractive index about 1.49. The solution running in the facility is carefully prepared by mixing Sodium-Iodine in distilled water to match the refractive index of the urethane-plastic trees. In this talk, experiments will be discussed in detail and measured velocity and turbulence statistics inside and above the canopy will be presented and discussed.

[1]This research is supported by NSF-AGS-1047550 and the Sardella Chair at Johns Hopkins University.

3:48PM L23.00002 On the routes to inertial mean dynamics in smooth- and rough-wall turbulent boundary layers[1]

JOSEPH KLEWICKI[2], FARAZ MEHDI, University of New Hampshire — Connections between the structure of smooth- and rough-wall turbulent boundary layers are established within the context of the order of magnitude properties exhibited by the terms in the mean momentum equation. These properties are shown to be associated with the processes by which inertial mean dynamics emerge with distance from the wall. A key element is the process by which the vorticity field becomes three-dimensional. In the smooth-wall case, vorticity stretching leads to the three-dimensionalization of the vorticity field in the region where the mean viscous force retains leading order. This underlies the well-established Reynolds number scaling behaviors exhibited by smooth-wall flows. Roughness modifies (generally augments) the process by which the vorticity field becomes three dimensional, rendering scalings for the route to inertial mean dynamics that depend on the relative scale separations between the inner, roughness, and outer scales. Evidence (from existing and recent experiments) of these combined scaling regimes is presented. The present analyses provide a basis for predicting where and physically why Townsend's similarity hypothesis should hold, as well as under what conditions outer similarity loses validity.

[1]The support of the ONR (N000140810836, grant monitor Ronald Joslin) is gratefully acknowledged.
[2]Also, University of Melbourne

4:01PM L23.00003 Investigation of Turbulence Modification by Momentum Injection Into Turbulent Flow Over a Rough Surface[1]

MARK MILLER, ALEXANDRE MARTIN, SEAN BAILEY, University of Kentucky — We present an experimental study conducted in a turbulent channel flow wind tunnel to determine the modifications made to the turbulent velocity spectrum by a sinusoidally rough, micro-perforated surface, both with and without flow injection through the surface. Preliminary results at moderate Reynolds numbers demonstrate that Townsend's hypotheseis is approximately valid at low momentum injection ratios. Whereas the magnitude and location of a near-wall peak in turbulence intensity remains largely unaffected by increasing flow injection, turbulence intensity in the logarithmic and outer layers increases and Townsend's hypothesis becomes invalid. Spectral analysis indicates that this increase in turbulence intensity reflects significant modifications made to the turbulence structure within these layers, even for very small injected momentum ratios. At high blowing rates, although the signature of very long wavelength motions persisted, the largest proportion of turbulent kinetic energy in the outer layer was found to be increasingly contained within turbulent scales corresponding to the thickness of the wall layer.

[1]This research is supported by Commonwealth of Kentucky funds in association with a NASA award entitled, Experimental Program to Stimulate Competitive Research, with NASA award identification number NNX10AV39A.

4:14PM L23.00004 Coherent structures and associated sub-grid scale energy transfer in a rough-wall turbulent channel flow[1]

JIARONG HONG, University of Minnesota, JOSEPH KATZ, CHARLES MENEVEAU, Johns Hopkins University, MICHAEL SCHULTZ, United States Naval Academy — The turbulence structure in a rough-wall channel flow and its role in subgrid scale (SGS) energy transfer are studied utilizing PIV data obtained in an optically index-matched facility. In streamwise-wall-normal $(x-y)$ planes, the averaged flow structure conditioned on high SGS flux exhibits a large inclined shear layer containing negative vorticity, bounded by an ejection below and a sweep above. The peaks of SGS flux and kinetic energy are spatially displaced from region of high-resolved TKE. In wall-parallel $x-z$planes, the conditional flow exhibits two pairs of counter-rotating vortices that induce a contracting flow at the SGS flux peak. Instantaneous realizations in the roughness sublayer confirm the presence of these vortex pairs at the intersection of two vortex trains. In the outer layer, the SGS flux peaks within isolated vortex trains that retain the roughness signature. To explain the planar signatures, we propose a flow consisting of U-shaped quasi-streamwise vortices that develop as spanwise vorticity is stretched in regions of high streamwise velocity between roughness elements. Flow induced by legs of adjacent U-shaped structures causes powerful ejections, which lift these vortices away from the wall. As a sweep is transported downstream, its interaction with the roughness generates a series of such events, leading to the formation of inclined vortex trains.

[1]Sponsored by ONR.

4:27PM L23.00005 Effect of Immersed Wall-Bounded Cylinders on Turbulent Boundary Layer Structure[1]

SHAOKAI ZHENG, ELLEN LONGMIRE, MICHAEL HALLBERG, MITCHELL RYAN, Aerospace Engineering and Mechanics, University of Minnesota — Single spanwise arrays of wall-mounted cylinders with $H/\delta \leq 0.2$, where H is the cylinder height and δ is the boundary layer thickness, were used to modify turbulent boundary layers (Re_τ=2500) in an attempt to affect the organization of the coherent structures in the logarithmic and outer regions. Flow downstream of several array spacings was investigated and compared against an unperturbed case. Instantaneous and averaged velocity fields in streamwise-spanwise planes were obtained by stereo PIV. The PIV cameras and laser sheet optics could be traversed at the local mean flow speed in order to track the evolution of larger structures in the flow. The results are analyzed to determine the streamwise evolution of dominant spanwise modes. Different array spacings are shown to either inhibit or reinforce the organization of vortex packet structures over streamwise distances up to 8δ. The flying stereo PIV measurements suggest also that dominant structures upstream of the arrays can strongly affect the organization and location of structures downstream.

[1]supported by NSF CBET-0933341

4:40PM L23.00006 Experimental test of the spectral analogue of the law of the wall in rough-pipe flows

CARLO ZUNIGA ZAMALLOA, GUSTAVO GIOIA, PINAKI CHAKRABORTY, Okinawa Institute of Science and Technology — We extend the recently proposed spectral analogue of Prandtl's law of the wall to obtain a scaling relation for the turbulent energy spectra in rough-pipe flows. To test this scaling relation we measure the streamwise component of the turbulent energy spectrum on numerous locations along the radii of three rough-walled pipes, for flows spanning a decade in Reynolds number. Our results are in excellent accord with the scaling relation.

4:53PM L23.00007 Measurement of turbulent flow upstream and downstream of a circular pipe bend , JUN SAKAKIBARA, NOBUTERU MACHIDA, Department of Engineering Mechanics and Energy, University of Tsukuba — We measured velocity distribution in cross sections of a fully developed turbulent pipe flow upstream and downstream of a 90-degree bend by synchronizing two sets of a particle image velocimetry (PIV) system. Unsteady undulation of Dean vortices formed downstream from the bend was characterized by the azimuthal position of the stagnation point found on the inner and outer sides of the bend. Linear stochastic estimation (LSE) was applied to capture the upstream flow field conditioned by the azimuthal location of the stagnation point downstream from the bend. When the inner-side stagnation point stayed below (above) the symmetry plane, the conditional streamwise velocity upstream from the bend exhibited high-speed streaks extended in a quasi-streamwise direction on the outer side of the curvature above (below) the symmetry plane. Similarity of the estimated structure and the very large scale motion (VLSM) will be presented.

5:06PM L23.00008 Evolution of turbulence characteristics from straight to curved pipes , GEORGE K. EL KHOURY, AZAD NOORANI, PHILIPP SCHLATTER, Linne FLOW Centre, KTH Mechanics, PAUL F. FISCHER, MCS, Argonne National Laboratory — Large-scale direct numerical simulations are performed to study turbulent flow in straight and bent pipes at four different Reynolds numbers: $Re_b = 5300$, 11700 (bent and straight) and 19000 and 37700 (only straight). We consider a pipe of radius R and axial length $25R$ with curvature parameter κ taken to be $0, 0.01$ and 0.1 for zero, mild and strong curvatures, respectively. The code used is Nek5000 based on the spectral element method. In the straight configuration, the obtained DNS data is carefully checked against other recent simulations, highlighting minute differences between the available data. Owing to a centrifugal instability mechanism, the flow in bent pipe ($\kappa \neq 0$) develops counter-rotating vortices, so-called Dean vortices. The presence of the secondary motion thus induces substantial asymmetries both in the mean flow and turbulence characteristics for the bent pipe. These asymmetries tend to damp turbulence along the inner side and correspondingly enhance it along the upper side. The results are validated with recent experiments, and we could confirm the peculiar behaviour of the friction factor for specific curvatures and Re, leading to a lower friction in curved pipes than in straight pipes for the same mass flux.

5:19PM L23.00009 Secondary motions induced by a 90° bend in turbulent pipe flow[1] , LEO HELLSTRÖM, ALEXANDER SMITS, Princeton University — Continuous stereoscopic particle image velocimetry (SPIV) was used to investigate the temporal behavior of curvature induced motions downstream a 90° bend in fully developed turbulent pipe flow. The velocity field was fully resolved in time for Reynolds numbers ranging from 1.3×10^4 to 3.6×10^4. Snapshot Proper Orthogonal Decomposition was performed on the data to extract the most energetic modes in the flow, which are believed to correctly identify the curvature induced secondary motions. These motions appear to be governed by a small number of highly energetic modes, active at different times. These modes may be used to reconstruct the flow, filtering the smaller structures, showing a "swirl switching" behavior, first noted by Tunstall & Harvey (1968).

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

Monday, November 19, 2012 3:35PM - 5:45PM –

Session L24 Aerodynamics IV 30E - Chair: James Hubner, University of Alabama

3:35PM L24.00001 The Role of Free Stream Turbulence on the Aerodynamic Performance of a Wind Turbine Blade , VICTOR MALDONADO, Texas Tech University, ADRIEN THORMANN, CHARLES MENEVEAU, Johns Hopkins University, LUCIANO CASTILLO, Texas Tech University, TURBULENCE GROUP COLLABORATION — In the present research, a 2-D wind turbine blade section based on the S809 airfoil was manufactured and tested at Johns Hopkins University in the Stanley Corrsin wind tunnel facility. A free stream velocity of 10 m/s produced a Reynolds number based on blade chord of 2.08.x10^5. Free stream turbulence was generated using an active grid placed 5.5 m upstream of the blade which generated a turbulence intensity, T_u of up to 6.1% and an integral length scale, L_∞ of about 0.15 m. The blade was pitched to a range of angles of attack, α from 0 to 18 degrees in order to study the effects of the integral length scales on the aerodynamic characteristics of the wind turbine under fully attached and separated flow conditions. Pressure measurements around the blade and wake velocity deficit measurements utilizing a hot-wire probe were acquired to compute the lift and drag coefficient. Results suggest that turbulence generally increases aerodynamic performance as measured by the lift to drag ratio, L/D except at 0 degrees angle of attack. A significant enhancement in L/D results with free stream turbulence at post-stall angles of attack of 16 and 18 degrees, where L/D increase from 2.49 to 5.43 and from 0.64 to 4.00 respectively. This is a consequence of delaying flow separation with turbulence (which is observed in the suction pressure distribution) which in turn reduces the momentum loss in the wake particularly at 18 degrees angle of attack.

3:48PM L24.00002 Investigation of the rotational flow effects in a pitching airfoil genetically optimized for vertical axis wind turbines , DANIELE RAGNI, TUDelft, LAURA VITALE, ANDREA IANIRO, Universita' Federico II, BEN GEURTS, CARLOS FERREIRA, TUDelft — In the present study, an airfoil optimized for vertical axis wind turbines applications has been developed with a genetic algorithm, selecting the geometry with maximum ($dc_l/d\alpha)/c_d$ among airfoils generated with 16 shape functions. The airfoil, operating in the curved trajectory of a vertical axis wind turbine, is usually optimized adopting conformal mappings in the straight path. Recent experimental results have shown disagreement with this approach, due to the forces determined in the curved flow path. To investigate the effects of flow rotation, an aluminum model (c=0.25m) has been manufactured from the optimized shape and further tested in the LST tunnel of the TUDelft at Reynolds number 10^6. Planar PIV experiments in combination with the PIV based load determination technique have been performed to simultaneously obtain velocity fields and loads. Results including velocity, pressure distributions, lift and drag are initially discussed in a steady airfoil configuration and compared with numerical results. Successively, the model has been unsteadily pitched using a magnetic linear actuator (up to 3 Hz frequency), with a free stream $V_\infty = 40$ m/s corresponding to Re = 0.7×10^6. Phase locked PIV vector fields have been acquired and compared to the steadily obtained results.

4:01PM L24.00003 High resolution velocimetry near the trailing edge of rigid and flexible airfoils undergoing unsteady motion[1] , DAVID OLSON, AHMED NAGUIB, MANOOCHEHR KOOCHESFAHANI, Michigan State University — The advantages of Molecular Tagging Velocimetry (MTV) are exploited in performing highly resolved measurements within the boundary layer, and downstream of the trailing edge of rigid and flexible NACA0012 airfoils undergoing canonical unsteady motions. Experiments are performed over a range of motion and flow parameters in an effort to establish the connections between airfoil motion trajectory, trailing edge flexure, and the time history of vorticity flux at the trailing edge. Specifically, multi-line MTV measurements, which are phase averaged relative to the airfoil motion, are used to examine the formation, evolution and characteristics of the wake vortices near the trailing edge and the concurrent behavior of the boundary layer immediately upstream. Results are used to gain insight into the effect of the trailing edge flexibility on the pattern of vorticity shed from, and the flow details around the trailing edge.

[1]This work was supported by AFOSR grant number FA9550-10-1-0342.

4:14PM L24.00004 Drag coefficient measurements of spheres with different surface patterns , HENDRIK HEISSELMANN, DANIEL STRUTZ, JOACHIM PEINKE, MICHAEL HOELLING, ForWind - Center for Wind Energy Research, Institute of Physics, University of Oldenburg — Precise drag force measurements of bluff bodies are an under-estimated challenge and in particular drag coefficients of bodies with rough surface structure are not very well documented in literature. In our contribution, we present a new setup for measurements of the acting drag forces on spheres and other bluff bodies. The examined bodies are attached to a slim supporting rod, which is held by thin steel wires in a cubical rigid frame, and the resulting velocity-dependent forces are measured by means of strain gauges. Besides a detailed description of the achieved experimental setup, we will present results from force measurements using smooth spheres and a sphere with a dimpled surface pattern. Measurements were performed for a Reynolds number range of 2,700 up to 230,000 under laminar inflow conditions as well as in turbulent flows generated by classical and fractal grids. An overview of the calculated drag coefficients will be given for different sphere types and for varying turbulence levels. The obtained results will be compared to those documented in literature.

4:27PM L24.00005 Comparison of Aerodynamic Coefficients from Low Aspect Ratio Membrane Wings and their Time-Averaged Shape[1] , NATHAN MARTIN, Rice University, ANDREW WRIST, ZHENG ZHANG, JAMES HUBNER, University of Alabama — Air flow over flexible membrane wings can induce vibration. The vibrating nature and the time-averaged curvature of a membrane wing may separately contribute to its improved aerodynamic characteristics compared to a flat plate of similar planform. To assess the effect of the time-averaged shape, a comparison of vibrating membrane wings and corresponding time-averaged shape of an aspect ratio two planform was conducted for membranes pre-tensions of 1%, 2%, and 4% and various cell aspect ratios. The membrane displacements were recorded using digital image correlation for each model at 6° and 18° angles of attack. The displacements were averaged, imported into CAD software, and printed using rapid prototyping equipment. The lift, drag, and pitching moment coefficients were acquired through wind tunnel testing at Reynolds number 50,000. The results indicate that membrane wings generate more drag but are more efficient than their time-averaged shapes due to greater lift.

[1]Funding received by NSF REU Grant 1062611 and AFOSR Grant FA9950-10-1-0152.

4:40PM L24.00006 Coupled Fluid and Structure Measurements over a Low Aspect Ratio Membrane Wing , LAWRENCE UKEILEY, MANUEL ARCE, AMORY TIMPE, University of Florida, ZHENG ZHANG, JAMES HUBNER, University of Alabama — The coupled effect of flow induced membrane deformations and their return influence on the flow are investigated on an aspect ratio 2 thin wing. The wings have multiple cells with a free scalloped trailing edge and are made by adhering heated, thin Silicone membranes to thin rectangular aluminum frames with a rigid leading edge and battens. Time-resolved flow and structure deformations are measured by synchronized acquisition of high-speed two-component particle image velocimetry (PIV) and stereoscopic digital image correlation (DIC) at a chord based Reynolds number of 48,000 and several angles of attack. Flow and structure metrics are compared for membrane with different values of pretension in the rubber. Instantaneous flow fields and mean flow properties are analyzed and compared to a rigid plate of the same dimensions. Specifically, the effects of membrane behavior on flow separation, shear layer size and location, along with vorticity will be analyzed. Power spectral density and correlation techniques are utilized, along with analysis of membrane mean deformation and rms fluctuation behavior to better understand the fluid-structure interactions and how the membranes interact with each other.

4:53PM L24.00007 Formulation for Time-resolved Aerodynamic Damping in Dynamic Stall[1] , THOMAS CORKE, University of Notre Dame, PATRICK BOWLES, United Technologies Research Center, DUSTY COLEMAN, FLINT THOMAS, University of Notre Dame — A new Hilbert transform formulation of the equation of motion for a pitching airfoil in a uniform stream yields a time resolved aerodynamic damping factor, $\Xi(t) = (\sqrt{(C_m^2(t) + \tilde{C}_m^2}/\alpha_{max}) \sin\psi(t)$, where $C_m(t)$ is the instantaneous pitch moment coefficient, and $\tilde{C}_m(t)$ is the Hilbert transform of $C_m(t)$, α_{max} is the pitching amplitude, and $\psi(t)$ is the time-resolved phase difference between the aerodynamic pitch moment and the instantaneous angle of attack. A $\Xi(t) < 0$ indicates unstable pressure loading that can be considered a necessary condition to excite stall flutter in an elastic airfoil. This will be illustrated in experiments with conditions producing "light" dynamic stall for a range of Mach numbers from 0.3-0.6. These reveal large negative excursions of $\Xi(t)$ during the pitch-up portion of the cycle that correlates with the formation and convection of the dynamic stall vortex. The fact that the cycle-integrated damping coefficient is positive in all these cases underscores how the traditional diagnostic masks much of the physics that underlies the destabilizing effect of the dynamic stall process. This new insight can explain instances of transient limit-cycle growth of helicopter rotor vibrations.

[1]Supported by Bell Helicopter

5:06PM L24.00008 Static versus dynamic stall development , KAREN MULLENERS, Leibniz Universitaet Hannover, MARKUS RAFFEL, German Aerospace Center, Goettingen, Germany — Stall on lifting surfaces is commonly encountered, mostly undesired, and occurs when a critical angle of attack is exceeded. Depending on the unsteady rate of change of the airfoil's angle of attack, static and dynamic stall are distinguished. To design efficient flow control measures, a fundamental understanding of the flow and vortex dynamics during stall development is desirable. Detailed information about the spatial and temporal evolution of the dominant flow features is obtained by time-resolved velocity field measurements combined with an extensive coherent structure analysis. Time-resolved flow field investigations during static and dynamic stall reveal flow reversal near the airfoil's surface at the beginning of stall development. At the interface between the region of reversed and free stream flow, a shear layer develops which plays the key role in the subsequent stall development. During dynamic stall, the shear layer rolls up into a large scale dynamic stall vortex which grows locally and temporally until vortex induced separation occurs. During static stall on the other hand, the shear layer rolls up continuously into large-scale structures that grow spatially.

5:19PM L24.00009 Large-Scale High-Resolution Cylinder Wake Measurements in a Wind Tunnel using Tomographic PIV with sCMOS Cameras[1] , DIRK MICHAELIS, LaVision GmbH, ANDREAS SCHROEDER, German Aerospace DLR — Tomographic PIV has triggered vivid activity, reflected in a large number of publications, covering both: development of the technique and a wide range of fluid dynamic experiments. Maturing of tomo PIV allows the application in medium to large scale wind tunnels. Limiting factor for wind tunnel application is the small size of the measurement volume, being typically about of 50 x 50 x 15 mm^3. Aim of this study is the optimization towards large measurement volumes and high spatial resolution performing cylinder wake measurements in a 1 meter wind tunnel. Main limiting factors for the volume size are the laser power and the camera sensitivity. So, a high power laser with 800 mJ per pulse is used together with low noise sCMOS cameras, mounted in forward scattering direction to gain intensity due to the Mie scattering characteristics. A mirror is used to bounce the light back, to have all cameras in forward scattering. Achievable particle density is growing with number of cameras, so eight cameras are used for a high spatial resolution. Optimizations lead to volume size of 230 x 200 x 52 mm^3 = 2392 cm^3, more than 60 times larger than previously. 281 x 323 x 68 vectors are calculated with spacing of 0.76 mm. The achieved measurement volume size and spatial resolution is regarded as a major step forward in the application of tomo PIV in wind tunnels.

[1]Supported by EU-project: no. 265695

5:32PM L24.00010 Transient Response of a Separated Flow over a Two-Dimensional Wing to a Short Duration Pulse[1] , DAVID WILLIAMS, Illinois Institute of Technology, THOMAS ALBRECHT, TOM WEIER, G. GERBETH, Helmholtz-Zentrum Dresden-Rossendorf Germany — A Lorentz force actuator located at the leading edge of a two-dimensional wing at 16 degrees angle of attack was used to introduce short-duration disturbances into a separated flow. The transient response of the separated region at Re = 10,000 was documented using time-resolved PIV measurements. The direction of the Lorentz force was changed between downstream and upstream directed disturbances, and details of the resulting flow field structures and lift measurements were studied. Saturation of the peak lift amplitude occurs as the actuation amplitude is increased from $0.0054 < C\mu < 0.21$ percent with the pulse duration fixed at 0.1 convective time. The effect of the pulse duration time on the lift response was examined using a fixed pulse amplitude, which showed that saturation occurred when pulse durations exceed 0.5 convective times. Differences in the coherent structures resulting from the upstream/downstream directed actuation were identified using the FTLE method. The initial development of the disturbed shear layer was strongly dependent on the direction of actuation, but the larger-scale separation did not show much difference. The relaxation of the separated region to the original flow state was essentially independent of the direction of actuation.

[1]Support by the Deutsche Forschungsgemeinschaft SFB 609 and AFOSR Grant FA9550-09-1-0189 is gratefully acknowledged.

Monday, November 19, 2012 3:35PM - 5:45PM –

Session L25 Flow Control: Boundary layer 31A - Chair: Clarence Rowley, Princeton University

3:35PM L25.00001 Numerical investigation of the AFRODITE transition control strategy[1] , SIMONE CAMARRI, Dipartimento di Ingegneria Aerospaziale - Università di Pisa, JENS H.M. FRANSSON, Linne Flow Centre, KTH Mechanics, ALESSANDRO TALAMELLI, DIEM, Alma Mater Studiorum - Università di Bologna-Forlì — The generation of properly distributed and shaped velocity streaks is a method to delay the Tollmien-Schlichting (TS) transition scenario in a boundary layer. Indeed, it is shown in the literature that stable velocity streaks in a Blasius boundary layer (BL) may lead to a damping of TS waves. This idea is explored in the AFRODITE[2] project, where streaks are generated experimentally in a Blasius BL by placing ad-hoc miniature vortex generators (MVGs) on the plate wall. In this presentation we show representative results obtained by the numerical setup that has been designed to support the experiments of the AFRODITE project. The DNSs are carried out using an open-source tool, Nek5000, which is a spectral-element code for incompressible flows. A 3D BL solver and its adjoint version are also used as additional numerical tools. The present DNSs include the simulation of the flow around the MVGs, so that the simulated streaks are those effectively generated by the considered devices. Details of the interactions between incoming TS waves and the MVGs are also investigated by DNS.

[1]CASPUR computing center (Roma, Italy) and the "C.M.Lerici" Foundation are gratefully acknowledged
[2]Advanced Fluid Research On Drag reduction In Turbulence Experiments, project funded by the European Research Council.

3:48PM L25.00002 Controller Selection and Placement in a Compressible Boundary Layer , DANIEL BODONY, MAHESH NATARAJAN, University of Illinois at Urbana-Champaign — A method for estimating the optimal location and type of flow control to use in compressible, turbulent flows is developed. Through linearizing the compressible Navier-Stokes equations about an unstable equilibrium point, the forward and adjoint equations of motion are used to estimate the structural sensitivity of the flow using a generalized "wavemaker" concept. Matrix optimization is used to enhance the structural sensitivity over possible controller types (e.g., mass or energy sources) and locations, with optimal solutions identifying advantageous control strategies. The algorithms and theory are applied to a high-subsonic separating boundary layer appearing in an S-duct and compared with more traditional optimal methods through adjoint-based gradient information. It is found that optimum locations for mass and energy sources are typically located upstream of those momentum sources, and use different physical mechanisms for affecting the flow.

4:01PM L25.00003 Exploring the Potential of Turbulent Flow Control Using Vertically Aligned Nanowire Arrays[1] , SEAN BAILEY, JOHN CALHOUN, CHRISTOPHER GUSKEY, MICHAEL SEIGLER, University of Kentucky, ANEESH KOKA, HENRY SODANO, University of Florida — We present evidence that turbulent flow can be influenced by oscillating nanowires. A substrate coated with vertically aligned nanowires was installed in the boundary wall of fully-developed turbulent channel flow, and the substrate was excited by a piezoceramic actuator to oscillate the nanowires. Because the nanowires are immersed in the viscous sublayer, it was previously unclear whether the small scale flow oscillations imparted into the bulk flow by the nanowires would influence the turbulent flow or be dissipated by the effects of viscosity. Our experiments demonstrated that the nanowires produced perturbations in the flow and contributed energy throughout the depth of the turbulent layer. A parallel investigation using a dynamically scaled surface of vertically aligned wires in laminar flow found that, even at low Reynolds numbers, significant momentum transport can be produced in the flow by the introduction of a travelling wave motion into the surface. These findings reflect the potential for using oscillating nanowires as a novel method of near-wall turbulent flow control.

[1]This work was supported by the Air Force Office of Scientific Research under FA9550-11-1-0140

4:14PM L25.00004 Effects of actuators and sensors on feedback control of transition in the 2D Blasius boundary layer , BRANDT BELSON, CLARENCE ROWLEY, Princeton University, ONOFRIO SEMERARO, KTH Royal Institute of Technology — We examine the effects of different types and positions of actuators and sensors on the performance and robustness of controllers designed to reduce the growth of Tollmien-Schlichting waves in the 2D Blasius boundary layer. We perform direct numerical simulations, and in order to facilitate controller design, we find reduced-order models with the Eigensystem Realization Algorithm. Good performance is obtained with the sensor upstream of the actuator, as in previous work. We categorize this as feedforward control, whereas sensors downstream of the actuator correspond to feedback control. The performance of feedforward controllers can be degraded by disturbances and perturbations, so we examine feedback controllers. The original choice of actuator and sensor results in ineffective feedback controllers (both for H2-optimal and simple classical controllers) due to weakly observable structures and a strict tradeoff between performance and robustness. Another choice of actuators and sensors is better suited for feedback control, and a PI feedback controller with the sensor slightly downstream of the actuator has good performance and robustness. Sensors farther downstream of the actuator cause inherent time delays that limit performance and robustness.

4:27PM L25.00005 Shock Wave Boundary Layer Interaction Control Using Pulsed DBD Plasma Actuators , ALEXANDRE LIKHANSKII, KRISTIAN BECKWITH, Tech-X Corporation — Flow separation in the shock wave boundary layer interaction (SWBLI) region significantly limits the development of supersonic inlets or scramjets. For past decades, scientists and engineers were looking for a way for active flow control of SWBLI. We will present our recent results of comprehensive simulations of SWBLI active control using pulsed nanosecond DBD plasma actuators at M=3. In the first part of simulations, we computed heat release from the ns pulse driven DBD plasma actuator to the flow using Tech-X plasma code Vorpal. This information has been consequently used in the simulations of SWBLI problem using Tech-X CFD code Nautilus. We compared baseline case with plasma actuators OFF to the case when plasma actuators were ON. We demonstrated strong perturbations in the region of SWBLI, suppression of flow separation and overall downstream increase of mass flow by ten percent when actuators are ON. We investigated the dependence of the results on the choice of different turbulence models and compared them to the laminar boundary layer case. We also performed parametric studies for different pulse repetition rates, pulse operation modes and DBD placement.

4:40PM L25.00006 Stability analysis of Boundary Layer in Poiseuille Flow through a modified Orr-Sommerfeld equation , JEAN BIO CHABI OROU, Université d'Abomey-Calavi, VINCENT MONWANOU, CLÉMENT MIWADINOU, None — For applications regarding transition prediction, wing design and control of boundary layers, the fundamental understanding of disturbance growth in the flat plate boundary layer is an important issue. In the present work we investigate the stability of boundary layer in Poiseuille flow. We normalize pressure and time by inertial and viscous effect. The disturbances are taken to be periodic in the spanwise direction and time. We present a set of linear governing equations for the parabolic evolution of wavelike disturbances. Then, we derive modified Orr-Sommerfeld equations that can be applied in the layer. We find that Squire's theorem is applicable for the boundary layer. We find also that normalization by inertial or viscous effects leads to the same stability or instability. We find through the graphs that transition from stability to instability or the opposite can occur according to the Reynolds number and the wave number.

4:53PM L25.00007 Time-resolved PIV of a turbulent boundary layer over a spanwise-oscillating surface[1] , KEVIN GOUDER, JONATHAN MORRISON, Imperial College — This work reports measurements of a turbulent boundary layer at $Re_\theta \approx 2500$, over a resonant spanwise-oscillating surface driven by a linear electromagnetic motor. Time-resolved PIV measurements of velocity are presented and supplemented by hot-wire measurements of velocity and direct drag measurements of friction drag using a drag balance. A maximum of 16% surface friction reduction, as calculated by the diminution of the wall-normal streamwise velocity gradient was obtained. The PIV laser beam was parallel to the plane of the oscillating surface at a height of $y^+ \approx 15$, hence, top-down views of the near-wall turbulence activity and the effect of the surface oscillation on its evolution were obtained. It has been shown that the imposition of a spanwise Stokes-like layer at a non-dimensional period of $T^+ = Tu_\tau^2/\nu \approx 100$ at peak-peak oscillation amplitudes equal to or larger than the mean streak spacing enabled the direct manipulation of the quasi-streamwise near-wall structures and caused fundamental changes in their evolution leading to reductions, for example, in the near-wall values of the mean-square of the streamwise fluctuating velocity component.

[1]This work was supported by Qinetiq, Airbus and EPSRC.

5:06PM L25.00008 Closed-Loop Control of Unsteady Transient Growth Disturbances in a Blasius Boundary Layer using DBD Plasma Actuators[1] , PHILIPPE LAVOIE, RONALD HANSON, University of Toronto, Institute for Aerospace Studies, KYLE BADE, AHMED NAGUIB, Michigan State University, Dept. of Mechanical Engineering, BRANDT BELSON, CLARENCE ROWLEY, Princeton University, Dept. of Mechanical and Aerospace Engineering — Plasma actuators have recently been shown to negate the effect of the transient growth instability occurring in a Blasius boundary layer for the purpose of delaying bypass transition. Specifically, during steady operation, the energy of a disturbance introduced via an array of static cylindrical roughness elements was reduced by up to 68%, as shown by Hanson et al (Exp. Fluids, 2010). In the present work, the actuators used in the aforementioned study were integrated into a complete closed-loop control system capable of negating unsteady transient growth disturbances induced in a Blasius boundary layer established in a wind tunnel. Shear stress measurements from an array of hot-wires mounted just above the surface of the boundary-layer plate downstream of the actuators are used to provide feedback information about the state of the boundary layer. The effectiveness and robustness of the closed-loop controller are rigorously established based on both control-model simulations and experiments.

[1]NSERC and NSF grant number: CMMI 0932546

5:19PM L25.00009 Numerical study of linear feedback control for form-drag reduction , JEREMY DAHAN[1], AIMEE MORGANS[2], Imperial College London — The present work is a numerical investigation of linear system identification and model-based feedback control methods for form-drag reduction. Large-Eddy Simulation (LES) is used to represent the flow over a simple bluff body with a sharp trailing edge, with a turbulent separation. For actuation, two types of perturbations are considered: a model of zero-net-mass-flux slot jets and momentum sources. Pressure measurements distributed over the base of the body provide the sensor information. The first part of the study will focus on the open-loop characterization of the flow. The base pressure field will be studied in relation to the wake dynamics. The effect of key actuation and flow parameters, such as actuation type, actuation location and Reynolds number, will be investigated. A black-box model of the flow response, obtained via system identification, will be examined. The second part will look at the design of robust controllers. It will be shown that uncertainties in the model and inflow conditions can be partially mitigated by the robustness of the controller. The behaviour of the feedback-controlled flow will be compared with the results achievable using open-loop forcing to draw conclusions about the success of the flow response model and the controller synthesis.

[1]PhD student in Department of Aeronautics
[2]Senior lecturer in Department of Aeronautics

5:32PM L25.00010 Linear modeling of turbulent skin-friction reduction due to spanwise wall motion , CARLOS DUQUE-DAZA[1], MIRZA BAIG, DUNCAN LOCKERBY, School of Engineering, University of Warwick, Coventry CV4 7AL, UK, SERGEI CHERNYSHENKO, Department of Aeronautics, Imperial College, London SW7 2AZ, UK, CHRISTOPHER DAVIES, School of Mathematics, Cardiff University, Cardiff CF24 4AG, UK, UNIVERSITY OF WARWICK TEAM, IMPERIAL COLLEGE TEAM, CARDIFF UNIVERSITY TEAM — We present a study on the effect of streamwise-travelling waves of spanwise wall velocity on the growth of near-wall turbulent streaks using a linearized formulation of the Navier-Stokes equations. The changes in streak amplification due to the travelling waves induced by the wall velocity are compared to published results of direct numerical simulation (DNS) predictions of the turbulent skin-friction reduction over a range of parameters; a clear correlation between these two sets of results is observed. Additional linearized simulations but at a much higher Reynolds numbers, more relevant to aerospace applications, produce results that show no marked differences to those obtained at low Reynolds number. It is also observed that a close correlation exists between DNS data of drag reduction and a very simple characteristic of the "generalized" Stokes layer generated by the streamwise-travelling waves.

[1]Carlos.Duque-Daza@warwick.ac.uk - School of Engineering, University of Warwick, Coventry CV4 7AL, UK caduqued@unal.edu.co - Department of Mechanical and Mechatronics Engineering, Universidad Nacional de Colombia.

Monday, November 19, 2012 3:35PM - 5:45PM –

Session L26 Reactive Flows VI: Fire, Soot, and Spray Combustion 31B - Chair: Kal Seshadri, University of California, San Diego

3:35PM L26.00001 Effects of buoyancy on heat transfer under an inclined flat plate , MICHAEL GOLLNER, University of Maryland, College Park, ANTONIO SANCHEZ, Universidad Carlos III de Madrid, FORMAN WILLIAMS, University of California, San Diego — A recent study of flame spread over and under a plastic fuel at different angles of inclination revealed new flame-spread behavior, where peak rates of flame spread were found on the underside of fuel surfaces, in contradiction with the traditional assumption that maximum spread rates occur in a vertical configuration (Gollner et al, Proc. Comb. Inst, 2012). Because flame spread is governed by heat transfer from flames to unignited fuel, a natural analogy can be drawn with heat transfer from an inclined, heated flat plate. Kierkus (IJHMT, 1968) performed a first-order perturbation analysis of this problem, however in taking the boussinesq approximation, the lack of density variation within the boundary layer resulted in no differences in the results between the under and over flat plate configurations. In this analysis, an attempt is made to perform a second-order perturbation analysis without invoking the boussinesq approximation, taking into account density differences within the boundary layer. These results are compared to heat-flux measurements made during flame spread both over-and-under inclined fuels to see if this observation is in fact caused by buoyancy effects within the boundary layer.

3:48PM L26.00002 A Numerical Study on Effects of Pressure and Gravity on Opposed Flow Flame Spread Rate over Thin Fuels , RANJIT SHUKLA, AMIT KUMAR, IIT Madras — In the recent years there has been renewed interest on effects of low pressure on combustion processes especially because of increased human endeavors in space. As access to space is expensive and so researchers have tried emulating effect of reduced gravity with reduced pressures at normal gravity. One such area of interest has been studies on spreading flames over condensed fuels. These studies are primarily driven by need of fire safety in low convection space environment. In quiescent space environment flame spread against the flow has been known to exist even where concurrent flame spread is not possible. Therefore, here in this work a 2D numerical model has been formulated to analyze the effects of pressure and gravity on flame spread behavior in an opposed flow configuration. An attempt is also made to arrive at pressure-gravity equivalence. The numerical model comprises of governing conservation equations for solid phase and gas phase. The 1D solid phase model (thin fuel) for ideally pyrolysing fuel is coupled to the gas phase by boundary conditions. Simulations carried over a range of gravity level from microgravity to the normal gravity and sub atmospheric pressures up to flame extinction show that the flame spread behavior and extinction is qualitatively the same from partial gravity of about 0.1g to 1g but quite different at near zero gravity. While the former is amenable to pressure-gravity equivalence modeling, the latter is not.

4:01PM L26.00003 Physics-based Modeling of Shrub Fires: Study of Distribution of Bulk Density and Moisture Content , AMBARISH DAHALE, BABAK SHOTORBAN, Department of Mechanical and Aeropspace Engineering, The University of Alabama in Huntsville, SHANKAR MAHALINGAM, College of Engineering, The University of Alabama in Huntsivlle — We utilized a physics-based model to investigate the influence of the spatial variation of solid-fuel bulk density and the solid fuel-moisture content on the behaviour of a shrub fire. The model accounts for the interaction of fluid dynamics, combustion of solid and gas phases, convective and radiative heat transfer, and thermal degradation of solid fuel. The turbulence was dealt with large eddy simulation and the gas-phase combustion was modeled through filtered flame surface density approach [Zhou & Mahalignam, Phys. Fluids, 2002]. Predictions from the model were compared against the experimental results, and fairly good agreement was observed between them. Vertical fire spread rate within the shrub and the time to initiate the ignition within the shrub were significantly affected by the spatial variation of the bulk density. They were also significantly influenced by the variation of the fuel moisture content. The amount of fuel burnt was also impacted by the change of fuel moisture content. The specific mechanisms responsible for the reduction in propagation speed in presence of higher bulk densities and/or moisture content were identified.

4:14PM L26.00004 ABSTRACT WITHDRAWN –

4:27PM L26.00005 Mathematical Modeling of Wildfire Dynamics[1] , KEVIN DEL BENE, DONALD DREW, Department of Mathematical Sciences, RPI — Wildfires have been a long-standing problem in today's society. In this paper, we derive and solve a fluid dynamics model to study a specific type of wildfire, namely, a two dimensional flow around a rising plume above a concentrated heat source, modeling a fire line. This flow assumes a narrow plume of hot gas rising and entraining the surrounding air. The surrounding air is assumed to have constant density and is irrotational far from the fire line. The flow outside the plume is described by a Biot-Savart integral with jump conditions across the position of the plume. The plume model describes the unsteady evolution of the mass, momentum, energy, and vorticity inside the plume, with sources derived to model mixing in the style of Morton, et al.[1956]. The fire is then modeled using a conservation derivation, allowing the fire to propagate, coupling back to the plume model. The results show that this model is capable of capturing the complex interaction of the plume with the surrounding air and fuel layer.

[1]Funded by NSF GRFP.

4:40PM L26.00006 Flamelet Radiation Modeling , JEFFREY DOOM, Minnesota State University, KRISHNAN MAHESH, University of Minnesota — A flamelet model is proposed that couples soot and radiation. The soot model from Carbonell et al. (*Combust. Flame.* 2009) is used. The radiation model is the P_1 gray and non–grey model from Modest (*Academic Press.* 2003) which are cast into the flamelet equations. A sooty ethylene flame is studied and a series of canonical calculations are performed. Results associated with the soot and radiation will be shown and compared to experiment.

4:53PM L26.00007 DNS of soot formation in three-dimensional turbulent non-premixed jet flames , ANTONIO ATTILI, FABRIZIO BISETTI, King Abdullah University of Science and Technology, MICHAEL E. MUELLER, Department of Mechanical and Aerospace Engineering, Princeton University, HEINZ PITSCH, RWTH Aachen University — A set of three-dimensional Direct Numerical Simulations (DNS) of soot formation in a three-dimensional n-heptane/air turbulent non-premixed jet flame has been performed to investigate the coupling between turbulence, chemistry, and soot dynamics with varying Damköhler number. Finite rate chemistry of Polycyclic Aromatic Hydrocarbons (PAH) is included in the chemistry model. Soot is described with a bivariate distribution in volume-surface sample space, and a selected number of moments of the distribution are transported via a recently proposed transport Lagrangian scheme. Closure of the soot moment equations is achieved via the Hybrid Method of Moments (HMOM). It is observed that, for smaller Damköhler number, the mass fraction of soot particles decreases while the number density stays approximately constant. In addition, Lagrangian statistics are used to study the evolution and transport of soot aggregates during their movement in physical and mixture fraction space.

5:06PM L26.00008 Validation of an LES Model for Soot Evolution against DNS Data in Turbulent Jet Flames , MICHAEL MUELLER, Princeton University — An integrated modeling approach for soot evolution in turbulent reacting flows is validated against three-dimensional Direct Numerical Simulation (DNS) data in a set of n-heptane nonpremixed temporal jet flames. As in the DNS study, the evolution of the soot population is described statistically with the Hybrid Method of Moments (HMOM). The oxidation of the fuel and formation of soot precursors are described with the Radiation Flamelet/Progress Variable (RFPV) model that includes an additional transport equation for Polycyclic Aromatic Hydrocarbons (PAH) to account for the slow chemistry governing these species. In addition, the small-scale interactions between soot, chemistry, and turbulence are described with a presumed subfilter PDF approach that accounts for the very large spatial intermittency characterizing soot in turbulent reacting flows. The DNS dataset includes flames at three different Damköhler numbers to study the influence of global mixing rates on the evolution of PAH and soot. In this work, the ability of the model to capture these trends quantitatively as Damköhler number varies is investigated. In order to reliably assess the LES approach, the LES is initialized from the filtered DNS data after an initial transitional period in an effort to minimize the hydrodynamic differences between the DNS and the LES.

5:19PM L26.00009 Direct numerical simulations of temporally developing turbulent reacting liquid-fueled jets , SHASHANK SHASHANK, HEINZ PITSCH, Stanford University — Liquid fueled engines are ubiquitous in the transportation industry because liquid fuel minimizes the weight and volume of propulsion systems. The combustion that occurs in these engines is an inherently multi-physics process, involving fuel evaporation, reaction kinetics, and high levels of turbulence. A desire for high fidelity data that explains complex interaction between different physical mechanisms motivates the consideration of direct numerical simulation (DNS) as an investigation tool. In this study three-dimensional DNS of a reacting n-heptane liquid fueled temporal jet have been performed to study auto-ignition and subsequent burning in conditions that are representative of a diesel engine environment. In these simulations the continuous phase is described using an Eulerian representation whereas Lagrangian particle tracking is used to model the dispersed phase. The results of this study will demonstrate the importance of unsteady effects, and of accounting for the interaction between different modes of combustion, when simulating spray combustion.

5:32PM L26.00010 Droplet evaporation and vapor mixing characteristics in a high-speed liquid jet spray , JUNJI SHINJO, Japan Aerospace Exploration Agency, AKIRA UMEMURA, Nagoya University — Droplet evaporation and vapor mixing in the early stage of a dense autoigniting spray are studied by detailed numerical simulation. Due to the relative velocity between droplets and air, heat and mass transfer is enhanced around the droplets. In the region of low droplet number density, the behavior is similar to that of a single droplet. In the region of high droplet number density, the interaction between neighboring droplets affects the transfer characteristics. Non-spherical geometry effect of droplets and ligaments will be also studied. The fuel/air mixture is formed non-uniformly due to the non-uniform droplet distribution and flow structure, which are determined during the spray formation. Reaction initiation is strongly affected by this mixture formation. Extension of temporal and spatial scales is finally sought for future effort in applying the results for real-scale combustors.

Monday, November 19, 2012 3:35PM - 5:32PM –

Session L27 Boundary-Layer Instability I 31C - Chair: Mujeeb Malik, NASA Langley Research Center

3:35PM L27.00001 Evolution of wavepacket over short compliant panels in a Blasius boundary layer , IGE BORI, K.S. YEO, National University of Singapore, HUA-SHU DOU, Zhejiang University of Science Technology, Hangzhou, China, XIJING ZHAO, National University of Singapore — Compliant surface has been proved in various theoretical studies as a promising tool in delaying transition. This study concerns our recent work carried on the evolution of pulse-initiated disturbance wavepackets over finite-length compliant panels in a Blasius boundary layer by direct numerical simulation (DNS) method. A finite section of the wall was replaced by a tensioned membrane on a damped foundation. By comparing with the rigid wall case, the upstream intervention by a finite compliant panel was found to effectively delay the onset of the incipient turbulent spot – an increase of about 51% in the transition distance with respect to the initiation point was obtained. Transition distance to the occurrence of the incipient turbulent spot was increased further to about 84% relative to a rigid wall when a second compliant panel was introduced. Spectral analysis shows the important role of the fundamental 2D modes in wavepacket evolution and the roles played by compliant panels in transition delay.

3:48PM L27.00002 Nonlinear intrinsic streaks in the flat plate boundary layer , CARLOS MARTEL, JUAN A. MARTIN, ETSI Aeronauticos, Universidad Politecnica de Madrid, 28040-Madrid — Luchini [JFM, vol.327, 1996] analyzed the flow near the leading edge of a flat plate boundary layer using the linearized problem around the Blasius solution. He found that there is just one single streaky mode (periodic in the spanwise direction) that grows downstream from the leading edge. The existence of this growing mode indicates that there is a one parameter family of 3D steady streak solutions that emerge from the leading edge of the boundary layer. In this presentation, we will numerically continuate downstream this family of intrinsic streaks (intrinsic because they appear in complete abscense of any free stream perturbations) using the Reduced Navier Stokes formulation, and we will show and comment the characteristics of the resulting Reynolds-number independent, fully nonlinear streaks.

4:01PM L27.00003 Dynamic mode decomposition of H-type transition to turbulence[1] , TARANEH SAYADI, JOSEPH NICHOLS, Center for Turbulence Research (CTR), Stanford, PETER SCHMID, LadHyX - CNRS - Ecole Polytechnique, PARVIZ MOIN, Center for Turbulence Research (CTR), Stanford — Dynamic mode decomposition (DMD) [1] is applied to a direct numerical simulation database of H-type transition to turbulence of a compressible, nominally-zero-pressure-gradient, spatially developing flat-plate boundary layer. The objective of this work is to identify the structures of dynamical importance throughout the transition region. DMD, viewed as an optimal phase averaging process in the context of the triple decomposition [2], is employed to assess the contribution of each coherent structure to the total Reynolds shear stress. In this region, it is observed that the total Reynolds shear stress gradient can be estimated accurately from only a few low-frequency DMD modes. These low-frequency modes are observed to correspond to the legs of hairpin vortices. Furthermore, DMD is applied to a large-eddy simulation (LES) database of the same configuration, generated using the dynamic Smagorinsky subgrid-scale model. The low-frequency DMD modes extracted from the LES, are, however, of lower amplitude than in the DNS, resulting in an underprediction of the Reynolds shear stress gradient and corresponding skin-friction coefficients.

[1] Schmid, P. J. JFM, 1-24, 2010.
[2] Reynolds, W. C. and Hussain, A. K. M. F. JFM, **54**, 236-288, 1972.

[1]Supported by the PSAAP program of DoE

4:14PM L27.00004 Self-sustained localized structures in a boundary-layer flow identified by edge tracking , DAN HENNINGSON, Linne FLOW Centre, KTH Mechanics, YOHANN DUGUET, LIMSI-CNRS, Paris, PHILIPP SCHLATTER, Linne FLOW Centre, KTH Mechanics, BRUNO ECKHARDT, Philipps-Universität Marburg — When a boundary layer starts to develop spatially over a flat plate, only disturbances of sufficiently large amplitude survive and trigger turbulence subcritically. Direct numerical simulation of the Blasius boundary-layer flow in a long and wide domain is carried out to track the dynamics in the region of phase space separating transitional from relaminarizing trajectories. In this intermediate regime, the corresponding disturbance is localized both in streamwise and spanwise directions, and spreads slowly in space. This structure is dominated by a robust pair of low-speed streaks, whose convective instabilities spawn hairpin vortices evolving downstream into transient disturbances. In contrast to previous work we find that the hairpin vortices are dynamically insignificant. A quasicyclic mechanism for the generation of offspring is unfolded using dynamical rescaling with the local boundary-layer thickness. The obtained quasi-cyclic character may be interpreted as an approach to an edge state in a spatially developing boundary layer. [PRL 108, 044501, 2012]

4:27PM L27.00005 Global stability and receptivity of swept attachment line boundary layer[1] , GIANLUCA MENEGHELLO, CNRS/Ecole Polytechnique, PETER SCHMID, PATRICK HUERRE, Ecole Polytechnique — The global stability and receptivity of the incompressible, viscous flow in the leading edge region of a swept wing is examined by solving the eigenvalues/eigenmodes problem associated with the Navier-Stokes operator linearized around a steady state base flow. A branch of eigenvalues is identified, which is associated with eigenvectors displaying a connection between attachment line and crossflow structures. The wavemaker region for these eigenvectors is shown to be close to the attachment line by computing the corresponding solution to the adjoint eigenvalue problem.

[1]The support of Airbus and the French National Center for Scientific Research (CNRS) is gratefully acknowledged.

4:40PM L27.00006 Effect of low freestream turbulence on crossflow instabiltiy , MOHAMMAD HOSSEINI, KTH Royal Institute of Technology, Department of Mechanics, Linné Flow Centre, SeRC, SE-10044 Stockholm, Sweden, ARDESHIR HANIFI, KTH Royal Institute of Technology, Dept of Mechanics, Linné Flow Centre, SeRC, SE-10044 Stockholm, Sweden, Swedish Defense Research Agency, FOI, DAN HENNINGSON, KTH Royal Institute of Technology, Department of Mechanics, Linné Flow Centre, SeRC, SE-10044 Stockholm, Sweden — The effect of freestream turbulence on the generation of crossflow disturbances in swept wings is investigated through direct numerical simulations. The set up follows the experimental set up provided by Hunt *et al.*[1] in their TAMU experiment. In this experiment the authors use ASU(67)-0315 wing geometry which promotes growth of crossflow disturbances. In this study, we fully reproduce the freestream isotropic homogenous turbulence through a DNS code using detailed freestream spectrum data provided by the experiment. The generated freestream fields are then applied as the inflow boundary condition for direct numerical simulation of the wing. The geometrical set up follows the experiment along with application of distributed roughness elements near the leading edge to precipitate stationary crossflow disturbances. Two different levels of freestream turbulence intensities are produced in order to study their effects on the initial amplitudes of the boundary layer perturbations. Additionally their influence on the transition location is examined.

[1]Hunt, L. E. 2011 Boundary-layer receptivity to three-dimensional roughness arrays on a swept-wing. PhD thesis, Texas A&M University.

4:53PM L27.00007 Linear Stability Analysis of Fully Three-Dimensional Boundary-Layers , WEI LIAO, National Institute of Aerospace, Hampton, VA 23666, MUJEEB MALIK, FEI LI, MEELAN CHOUDHARI, CHAU-LYAN CHANG, NASA Langley Research Center, Hampton, VA,23681 — Stability and transition of three-dimensional (3D) finite-span swept-wing boundary layers is of interest here. It is common practice to use quasi-3D boundary-layer codes for generating mean flows for stability analysis of swept-wing flows because of their efficiency and simplicity. However, the use of infinite span approximation or the spanwise conical flow assumption in these codes becomes questionable for fully three-dimensional boundary layers. In this work, the results of stability analysis based on mean flows generated by a quasi-3D boundary layer solver and an unstructured-grid Navier-Stokes solver (FUN3D[1]) are compared for a swept wing-glove assembly.[2,3] The N-factor evolution based on the full-Navier-Stokes computation is shown to differ significantly from that based on the quasi-3D boundary layer codes owing to the un-sweep of the isobars caused by the limited glove span. This points to the need for stability analysis based on Navier-Stokes solutions or possibly fully 3D boundary layer codes when the underlying flow develops strong three-dimensionality. The substantial reduction in maximum N factor (from about 20 to 12 for the case studied here) also indicates the possibility of stabilizing crossflow instability by using fully 3D design methods.

[1]http://fun3d.larc.nasa.gov/
[2]Belisle et al., AIAA-2012-2667.
[3]Liao et al., AIAA-2012-2690.

5:06PM L27.00008 Effect of Surface Imperfections and Excrescences on the Crossflow Instability[1] , MATTHEW TUFTS, GLEN DUNCAN, JR., BRIAN CRAWFORD, HELEN REED, WILLIAM SARIC, Texas A&M University — Presented is analysis of the planned SWIFTER experiment to be flown on Texas A&M University's O-2A aircraft. Simultaneous control of the crossflow and streamwise boundary-layer instabilities is a challenge for laminar flow control on swept wings. Solving this problem is an active area of research, with a specific need to quantify the effect of surface imperfections and outer mold line excrescences on crossflow instabilities. The SWIFTER test article is a modification of a prior-tested flight model, with the additional capability of creating controlled excrescences in flight. Using a finite-element Navier-Stokes solution and a spectrally accurate boundary-layer solver, coupled with linear and nonlinear stability analyses, we show that the flow field over the test article is well suited to this study. Results are compared with flight data.

[1]The work is supported by the Air Force Research Laboratory through General Dynamics Information Technology, Inc. under sub Agreement No USAF-3446-11-50-SC-01 and the Texas A&M Supercomputing Facility.

5:19PM L27.00009 Computations of Crossflow Transition in Supersonic Swept Wing Boundary Layers , LIAN DUAN, National Institute of Aerospace, Hampton, VA, MEELAN CHOUDHARI, FEI LI, NASA Langley Research Center, Hampton, VA, MINWEI WU, National Institute of Aerospace, Hampton, VA — A common cause for transition over swept wing configurations of a supersonic aircraft is the crossflow instability of the three-dimensional boundary layer flow. This study seeks to analyze the nonlinear stages of transition due to crossflow instability, with the eventual goals of enabling efficient yet accurate predictive models and, potentially, obtaining clues for better control of the transition process. To achieve these goals, direct numerical simulations are performed to examine the laminar breakdown process in a supersonic swept airfoil boundary layer. Different mechanisms of transition are studied with an emphasis on the breakdown initiated by the high-frequency secondary instability of stationary crossflow modes. The secondary instability is introduced via inflow forcing derived from a two-dimensional, partial-differential-equation based eigenvalue computation. The simulation tracks the linear and nonlinear growth of the secondary instability wave, the resulting onset of laminar-turbulent transition, and the fully turbulent flow downstream. As the secondary instability grows, small rib-like structures develop on top of the tubular structure. These structures are aligned at an oblique angle to the axis of the crossflow vortex, and are similar to the rib-like structures observed in previously reported computations of secondary instability in incompressible flows. Farther downstream, even smaller structures emerge and the laminar breakdown process ensues.

Monday, November 19, 2012 3:35PM - 5:19PM –

Session L28 Industrial Applications I: Turbines and Engines 32A - Chair: John Cimbala, The Pennsylvania State University

3:35PM L28.00001 POD- Mapping and analysis of hydroturbine exit flow dynamics[1] , MORTEN KJELDSEN, Flow Design Bureau AS, PAL HENRIK FINSTAD, NTNU — Pairwise radial dynamic measurements of the swirling draft tube flow have been made at the 25 MW Svorka power plant in Surnadal operating at 48% load at 6 radial and 7 angular positions. The data is analyzed with traditional methods as well as with POD. The measurements were made in the turbine draft tube/exit flow in an axial measurement plane about 1200mm downstream the turbine runner. The draft tube diameter in the measurement plane is about 1300mm. The flow rate during measurements was close to 5.8m3/s. Two probes were used; both of length Le=700 mm and made of stainless steel with an outer diameter of Do=20 mm and inner diameter Di=4mm. At the end of each probe a full bridge cylindrical KULITE xcl152, 0-3.5, was mounted. 90 seconds samples at 10 kS/s were taken. The POD analysis largely follows that of Tutkun et al (see e.g. AIAA J., 45,5,2008). The analysis shows that 26% of the pressure pulsation energy can be addressed to azimuthal mode 1.

[1]The work has been supported by Energy Norway.

3:48PM L28.00002 Caution: Precision Error in Blade Alignment Results in Faulty Unsteady CFD Simulation[1] , BRYAN LEWIS, JOHN CIMBALA, ALEX WOUDEN, The Pennsilvania State University — Turbomachinery components experience unsteady loads at several frequencies. The rotor frequency corresponds to the time for one rotor blade to rotate between two stator vanes, and is normally dominant for rotor torque oscillations. The guide vane frequency corresponds to the time for two rotor blades to pass by one guide vane. The machine frequency corresponds to the machine RPM. Oscillations at the machine frequency are always present due to minor blade misalignments and imperfections resulting from manufacturing defects. However, machine frequency oscillations should not be present in CFD simulations if the mesh is free of both blade misalignment and surface imperfections. The flow through a Francis hydroturbine was modeled with unsteady Reynolds-Averaged Navier-Stokes (URANS) CFD simulations and a dynamic rotating grid. Spectral analysis of the unsteady torque on the rotor blades revealed a large component at the machine frequency. Close examination showed that one blade was displaced by 0.0001° due to round-off errors during mesh generation. A second mesh without blade misalignment was then created. Subsequently, large machine frequency oscillations were not observed for this mesh. These results highlight the effect of minor geometry imperfections on CFD solutions.

[1]This research was supported by a grant from the DoE and a National Defense Science and Engineering Graduate Fellowship.

4:01PM L28.00003 Effect of Marine Hydrokinetic array configuration on power extraction , MICHAEL VOLPE, MARIA LAURA BENINATI, Bucknell University, MICHAEL KRANE, ARNIE FONTAINE, Pennsylvania State University — Experiments are presented to explore how the spatial arrangement of a Marine Hydrokinetic (MHK) turbine array affects the power extracted from an individual turbine. The experiments were performed in the small-scale testing platform in a hydraulic flume facility (9.8 m long, 1.2 m wide and 0.4 m deep) at Bucknell University. The study focuses on a V-shaped sub-array, with the vertex element in the downstream location. The vertex element is a two-bladed horizontal axis turbine, is loaded using a metal-brush motor, and the two upstream elements are circular perforated disks designed to mimic the time-averaged flow disturbance imparted to the flow by a similar turbine. Experiments were performed for a range of stream-wise separation between the perforated plate elements and the turbine element. For each separation distance, drag on the turbine, power extracted by the turbine, as well as the velocity field upstream and downstream of the turbine, were all measured. From these measurements, the flow incident on the turbine and the rate of work done by the flow on the turbine are obtained. Results show that placing a turbine in the accelerated flow region between the wakes of upstream array elements can result in increased energy extraction.

4:14PM L28.00004 Numerical modeling of the effects of a free surface on the operating characteristics of Marine Hydrokinetic Turbines[1] , SAMANTHA ADAMSKI, ALBERTO ALISEDA, University of Washington — Marine Hydrokinetic (MHK) turbines are a growing area of research in the renewable energy field because tidal currents are a highly predictable clean energy source. The presence of a free surface may influence the flow around the turbine and in the wake, critically affecting turbine performance and environmental effects through modification of wake physical variables. The characteristic Froude number that control these processes is still a matter of controversy, with the channel depth and turbine's depth, blade tip depth and diameter as potential candidates for a length scale used in literature. We use the Volume of Fluid model to track the free surface dynamics in a RANS simulation with a BEMT model of the turbine to understand the physics of the wake-free surface interactions. Pressure and flow rate boundary conditions for channel's inlet, outlet and air side have been tested in an effort to determine the optimum set of simulation conditions for MHK turbines in rivers or estuaries. Stability and accuracy in terms of power extraction and kinetic and potential energy budgets are considered. The goal of this research is to determine, quantitatively in non dimensional parameter space, the limit between negligible and significant free surface effects on MHK turbine analysis.

[1]Supported by DOE through the National Northwest Marine Renewable Energy Center

4:27PM L28.00005 Experimental study of the lift and drag characteristics of a cascade of flat plates in a configuration of interest for tidal energy converters[1] , FAICAL FEDOUL, LUIS PARRAS, CARLOS DEL PINO, RAMON FERNANDEZ-FERIA, University of Malaga (Spain) — Wind tunnel experiments are conducted for the flow around both a single flat plate and a cascade of three parallel flat plates at different angles of incidence to compare their lift and drag coefficients in a range of Reynolds number about 10^5, and for two values of the aspect ratio of the flat plates. The selected cascade configuration is of interest for a particular type of tidal energy converter. The lift and drag characteristics of the central plate in the cascade are compared to those of the isolated plate, finding that there exist an angle of incidence, which depends on the Reynolds number and the aspect ratio, above which the effective lift of the plate in the cascade becomes larger than that of an isolated plate. These experimental results, which are also analyzed in the light of theoretical predictions, are used as a guide for the design of the optimum configuration of the cascade which extracts the maximum power from a tidal current for a given value of the Reynolds number.

[1]Supported by the Ministerio de Ciencia e Innovacion (Spain) Grant no. ENE2010-16851.

4:40PM L28.00006 RANS simulations of a flow over a rotating disk: grid sensitivity analysis , SVETLANA V. POROSEVA, MICHAEL A. SNIDER, University of New Mexico — In industry, a need in accurate and reliable flow simulations often comes with the requirements for reduced computational time and cost. To find an optimal balance between these requirements, a sensitivity analysis of simulation results with respect to various simulations parameters should be conducted at the early stage of computations. When experimental data is not available for validating simulations in a given flow geometry, such study can be conducted for a relevant benchmark problem instead. In the current study, a sensitivity analysis was conducted for flow simulations over a rotating disk. This case can serve as a benchmark problem for flow simulations around a wind turbine, for example. Indeed, as a number of blades on a wind turbine approaches infinity, the turbine's geometry transforms to a solid disk. The convergence of simulation results with respect to the size of computational domain, boundary proximity, grid stretching, and initial grid wall spacing was analyzed for five standard turbulence models available in Star-CCM+ software. Objectives were i) to find the coarsest grid that closely reproduces results obtained with a given turbulence model on more fine grids and ii) to find a model that is more robust to changes in the gird parameters.

4:53PM L28.00007 Evaluating Fuel-Air Mixing in a Direct-Injection Hydrogen-Fueled Internal Combustion Engine , ALIREZA EBADI, CHRISTOPHER WHITE, University of New Hampshire — Proper orthogonal decomposition (POD) is used to decompose in-cylinder particle image velocimetry (PIV) vector fields acquired in a direct injection hydrogen-fueled internal combustion engine (DI-H2ICE) into mean, coherent, and incoherent vector fields, where the coherent vector fields are presumed to capture the cycle-variability of the flow. The POD vector fields are then used to investigate the effects of fuel injection timing on in-cylinder turbulence and fuel-air mixing.

5:06PM L28.00008 Swirling flow in model of large two-stroke diesel engine , K.E. MEYER, K.M. INGVORSEN, Department of Mechanical Engineering, Technical University of Denmark, Building 403, DK-2800 Kgs. Lyngby, Denmark, S. MAYER, MAN Diesel & Turbo, Teglholmsgade 41, DK-2450 Copenhagen SV, Denmark, J.H. WALTHER[1], Department of Mechanical Engineering, Technical University of Denmark, Building 403, DK-2800 Kgs. Lyngby, Denmark — In large two-stroke uniflow scavenged marine diesel engines fresh air is blown in through angled ports in the bottom of the cylinder liner forcing the burned gas out through an exhaust valve in the cylinder head. The scavenging flow is a transient (opening/closing ports) confined port-generated turbulent swirling flow, with complex phenomena such as central recirculation zones, vortex breakdown and vortex precession. A scale model of a simplified cylinder is created with a transparent cylinder five diameters long. The flow in the experiment has a Reynolds number of 50,000 based on the cylinder diameter and bulk velocity. Stereoscopic Particle Image Velocimetry (PIV) is used to investigate the flow for cases with both static and moving piston. Port angles of 0, 10, 20 and 30 degrees are considered. Although the flow has a relatively low swirl number of around 0.4, a central recirculation zone is observed indicating a vortex breakdown. The steady flow is analyzed with proper orthogonal decomposition revealing systematic variations in the shape and location of the vortex core. Transient measurements using phase-locked PIV are carried out with moving piston. The transient measurements reveal a sudden rapid change in flow topology as a central recirculation zone is formed.

[1]Also at: Computational Science and Engineering Laboratory, ETH Zurich, Universitatsstrasse 6, CH-8092 Zurich, Switzerland

Monday, November 19, 2012 3:35PM - 5:32PM –

Session L29 Chaos, Fractals, and Dynamical Systems I: Lagrangian Coherent Structures 32B - Chair: Wenbo Tang, Arizona State University

3:35PM L29.00001 Lagrangian Coherent Structures separate dynamically distinct regions[1] , DOUGLAS KELLEY, Department of Materials Science & Engineering, Massachusetts Institute of Technology, MICHAEL ALLSHOUSE, Department of Mechanical Engineering, Massachusetts Institute of Technology, NICHOLAS OUELLETTE, Department of Mechanical Engineering & Materials Science, Yale University — Lagrangian Coherent Structures (LCS) are special material lines that play a role in unsteady flow analogous to the stable and unstable manifolds of hyperbolic fixed points in periodic flows. Since they are material lines, fluid elements cannot cross them, and thus they separate regions of the flow field that are kinematically distinct. Using recently developed filter-space techniques that allow us to localize spectral transport processes in space, we study the Lagrangian averages of scale-to-scale energy transfer in an experimental quasi-two-dimensional flow. We find that, surprisingly, LCS appear to divide regions that are dynamically as well as kinematically distinct. We find that on the average LCS separate parts of the flow field with coherent (in a Lagrangian sense) scale-to-scale energy fluxes in different directions.

[1]This work was supported by the U.S. National Science Foundation under Grant No. DMR-0906245.

3:48PM L29.00002 Particle manipulation using vibrating cilia , PHANINDRA TALLAPRAGADA, SCOTT KELLY, University of North Carolina, Charlotte — The ability to manipulate small particles suspended in fluids has many practical applications, ranging from the mechanical testing of macromolecules like DNA to the controlled abrasion of brittle surfaces for precision polishing. A natural method is non-contact manipulation of particles through boundary excitations. Particle-manipulation via a vibrating cilia to establish controlled fluid flows with desired patterns of transport is one such bioinspired method. We show experimental results on the clustering and transport of finite-sized particles in the streaming flow set up by the oscillating cilia. We further show computations to explain the effects of hyperbolic structures in the four dimensional phase space of the dynamics of finite-sized particles.

4:01PM L29.00003 Passive scalar statistics and its dependence on Lagrangian coherent structures in stochastic flows[1] , WENBO TANG, PHILLIP WALKER, Arizona State University, MICHAEL ALLSHOUSE, Massachusetts Institute of Technology, DIEGO DEL-CASTILLO-NEGRETE, Oak Ridge National Lab — In recent years, various mathematical tools have been developed to identify the organizing mixing patterns in deterministic aperiodic dynamical systems. In this talk we will discuss the dependence on different identification methods, (Lagrangian Okubo-Weiss, Finite-time Lyapunov exponents, ergodicity partition and geodesic theory), of Lagrangian statistics associated with stochastic aperiodic dynamical systems (e.g. fluid flows with subgrid-scale uncertainties). Gaussian and Lévy type noises will be considered.

[1]Thanks to: NSF DMS-1212144

4:14PM L29.00004 Finite-time statistics of scalar diffusion in Lagrangian coherent structures[1] , PHILLIP WALKER, WENBO TANG, Arizona State University — When investigating chaotic mixing in nonlinear aperiodic dynamic systems, the domain can be frame-independently partitioned into different regions identified by Lagrangian coherent structures (LCS). We consider stochastic scalar dispersion associated with LCS and find that the statistics of various moments exhibit strong coherence in separate flow partitions. The probability density of dispersion approach self-similar profiles with anomalous exponents at intermediate time scales. Such coherence in statistics indicate that the Lagrangian topology highlight variability of diffusion. In this talk we explore such correlation between Lagrangian topology, as identified by LCS, and effective mixing.

[1]Thanks to: NSF DMS-1212144

4:27PM L29.00005 Efficient and robust detection of transport barriers using the geodesic approach , MICHAEL ALLSHOUSE, Massachusetts Institute of Technology, JEAN-LUC THIFFEAULT, University of Wisconsin - Madison, THOMAS PEACOCK, Massachusetts Institute of Technology — There is an increasing number of applications where the identification of transport barriers is valuable. A recent advance in transport barrier theory has created a unified approach to detecting hyperbolic (LCS based), elliptic (KAM based), and parabolic (shear jets) transport barriers. We have developed an algorithm building on the details from this result. We present a number of modifications to the algorithm which aim to increase accuracy, reduce unnecessary calculations, and make the method suitable for practical applications. This approach is then applied to an ocean surface model to study transport barriers present during the Deepwater Horizon spill.

4:40PM L29.00006 Integrated computation of Lagrangian coherent structures during DNS of unsteady and turbulent flows[1] , JUSTIN FINN, SOURABH APTE, Oregon State University — The computation of Lagrangian coherent structures (LCS) typically involves post processing of experimentally or numerically obtained fluid velocity fields to obtain the finite time Lyapunov exponent (FTLE) via a sequence of flow maps (vector fields which describe fluid displacement patterns over a finite time interval, $t_0 \pm T$). However, this procedure can be prohibitively expensive for large-scale complex flows of engineering interest. In this work, an alternative approach involving computation of the FTLE on the fly during direct numerical simulation (DNS) of the 3D Navier-Stokes equations is developed. This incorporation of the FTLE computations into a parallel DNS solver relies on Lagrangian particle tracking to compose forward time flow maps, and an Eulerian treatment of the backward time flow map [Leung, J. Comp. Physics 2011] coupled with a semi-Lagrangian advection scheme. The time T flow maps are accurately constructed from smaller sub-steps [Brunton & Rowley, Chaos 2010], resulting in low CPU and memory requirements for computing evolving FTLE fields. Illustrative examples will be presented to demonstrate the capability of the approach including the evolution of a turbulent vortex ring and turbulent flows in complex porous media.

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

4:53PM L29.00007 Visualization of invariant sets in incompressible fluid flows from Lagrangian data , MARKO BUDIŠIĆ, IGOR MEZIĆ, University of California - Santa Barbara — We analyze Lagrangian data of incompressible fluid flows to provide a coarse-grained visualization of material transport. It is often difficult to resolve features in 3D plots of trajectories. Instead, we visualize sets that are transport-invariant. In spirit, the algorithm groups trajectories that are similar *on average* into invariant sets of different spatial scales. Invariant sets are represented by level-sets of flow-invariant functions, which partition the space. We construct such invariant functions by averaging a basis set, e.g., Fourier basis, along Lagrangian trajectories. Our Ergodic Quotient algorithm then combines trajectory averages to form scale-ordered invariant partitions. The lower orders visualize coarse features, e.g., dominant vortices, while higher orders resolve sub-features, e.g., secondary vortices weaving around the dominant ones. The algorithm is suitable for visualization of both numerical and experimental Lagrangian data. It has a benefit of not requiring an access to the entire space: it is possible to resolve the features even by seeding initial conditions into experimentally accessible regions and allowing for the flow to disperse the tracers. We demonstrate the algorithm on several numerical flows and explain future extensions.

5:06PM L29.00008 Short- and Long- Time Transport Structures in a Three Dimensional Time Dependent Flow[1] , RODOLPHE CHABREYRIE, STEFAN LLEWELLYN SMITH, University of California, San Diego Mechanical and Aerospace Engineering — Lagrangian transport structures for three-dimensional and time-dependent fluid flows are of great interest in numerous applications, particularly for geophysical or oceanic flows. In such flows, chaotic transport and mixing can play important environmental and ecological roles, for examples in pollution spills or plankton migration. In such flows, where simulations or observations are typically available only over a short time, understanding the difference between short-time and long-time transport structures is critical. In this talk, we use a set of classical (i.e. Poincaré section, Lyapunov exponent) and alternative (i.e. finite time Lyapunov exponent, Lagrangian coherent structures) tools from dynamical systems theory that analyze chaotic transport both qualitatively and quantitatively. With this set of tools we are able to reveal, identify and highlight differences between short- and long-time transport structures inside a flow composed of a primary horizontal contra-rotating vortex chain, small lateral oscillations and a weak Ekman pumping. The difference is mainly the existence of regular or extremely slowly developing chaotic regions that are only present at short time.

[1]This research was funded by the ONR MURI Dynamical Systems Theory and Lagrangian Data Assimilation in 3D+1 Geophysical Fluid Dynamics.

5:19PM L29.00009 The role of filamentation and vortex merging in coastal particle accumulation and transport , CHERYL HARRISON, University of California Santa Cruz, DAVID SIEGEL, University of California Santa Barbara, SATOSHI MITARAI, Okinawa Institute of Science and Technology — Understanding ocean transport of coastally released material is crucial for predicting planktonic and pollutant transport. Here we use a coupled particle-tracking/ocean circulation model of an upwelling current to identify important transport processes at meso- to submesoscales. Buoyant particles released over the continental shelf simulate surface following planktonic material. Particles are largely organized into filaments found between mesoscale eddies that correspond to attracting Lagrangian coherent structures (LCS), material curves that map filamentation and transport boundaries, and here correlate with temperature fronts and their associated secondary ageostrophic circulation. Filamentation and vortex merging reduce mixing, aggregating particles from many source regions and release times into small, highly dense packets. As predicted by structural stability of LCS, filaments and packets are robust to strong levels of random walk "swimming" perturbations, indicating these processes will be robust to a wide range of planktonic behavioral strategies. This study demonstrates that 1) coherent flow structures play an important role in pelagic transport of marine propagules, plankton and floating pollutants in the coastal ocean, and 2) dynamical systems techniques will have broad applicability in these systems.

Monday, November 19, 2012 3:35PM - 5:45PM –

Session L30 Fluid Education 33A - Chair: Frank Jacobitz, University of San Diego

3:35PM L30.00001 The Fluid Dynamics Demo Kit: Part I[1] , KAREN FLACK, United States Naval Academy, PATRICK UNDERHILL, Rensselaer Polytechnic Institute, KATHY PRESTRIDGE, Los Alamos National Laboratory — The goal of this project is to develop a fluid dynamics demonstration/experiment kit that can be used by professors and graduate students at high school outreach events. The demonstrations in the kit will be easy to use and true crowd pleasers in order to inspire understanding and pique curiosity about the physics of flow. The kits will be inexpensive, containing readily available materials so that teachers can duplicate the demonstrations and experiments. The kits will be left with the teachers as a gift from the American Physics Society. The experiments and demonstrations cover the concepts of conservation of mass, momentum, and energy, Bernoulli's equation, frictional losses and the ideal gas law. For each experiment, the teachers will receive presentation material, access to instructional videos, plus a worksheet that can be used in a high school physics classroom. This kit has been developed through the efforts of the APS-DFD Mentoring and Outreach Committee and has received funding from the APS-DFD.

[1]Work funded by the APS-DFD.

3:48PM L30.00002 The Fluid Dynamics Demo Kit: Part II , PATRICK UNDERHILL, HANNAH FIX, Rensselaer Polytechnic Institute, TIMOTHY HAINES, University of Washington, KATHY PRESTRIDGE, Los Alamos National Laboratory, KAREN FLACK, United States Naval Academy — This talk will focus on the current contents and efforts building a fluid dynamics demonstration/experiment kit through the APS-DFD Mentoring and Outreach Committee and with funding from the APS DFD. The current experiments include predicting how far the water from a water gun will go, predicting how long a Heron's fountain will run, measuring the viscosity of corn syrup by measuring terminal velocity, and measuring the flow rate through a siphon. We will discuss the testing and development of these experiments and the results of field testing. Ideas for future experiments will also be discussed.

4:01PM L30.00003 Water bottle rocket in undergraduate laboratory , WILLIAM SCHULTZ, University of Michigan — In the winter semester of 2012, we implemented the modeling and testing of a water bottle rocket in ME 495, the Senior Laboratory in Mechanical Engineering at the University of Michigan. The four week lab was the most well received by the students in recent memory. There were significant challenges, but the result was a thorough review of their undergraduate fluids class with some advanced concepts such as directional stability of a projectile. The student teams designed their own rockets based on one of many standard 20 ounce soft drink bottles. The culminating contest brought impressive results and a surprise ending.

4:14PM L30.00004 Fluid Mechanics: The Pamphlet , EVAN VARIANO, UC Berkeley — One impediment to student learning in introductory fluid mechanics courses is that the fundamental laws of physics can become lost in the "noise" of dozens of semi-empirical equations describing special cases. This can be exacerbated by trends in textbooks and other teaching media. This talk will explore a minimalist approach, whereby the entire content of introductory fluids is distilled to a single 1-page pamphlet, designed to emphasize the governing equations and their near-universal applicability. We are particularly interested in hearing feedback from the audience on ways to further distill the content while keeping it accessible and useful. To further emphasize the difference between the fundamental laws and the many specific cases, we have begun assembling a complementary resource: a field guide to fluid phenomena, which mixes the approach of Van Dyke's book with a standard field guide. This is designed to emphasize that there is a "zoology" of fluid phenomena, to which the same small set of fundamental laws has been applied repeatedly. These materials may be useful in helping AP Physics teachers cover fluid mechanics, which is an under-utilized opportunity to introduce young scientists to our field of study.

4:27PM L30.00005 Experience revising an advanced-undergraduate/beginning-graduate fluid mechanics textbook , DAVID DOWLING, Dept. of Mech. Eng., University of Michigan — In the fall of 2009, Elsevier Inc. approached me about taking over as the lead author of the fluid mechanics textbook by P. K. Kundu and I. M. Cohen. I subsequently agreed and this presentation provides the story of the process and the approach taken to revising this fluid mechanics textbook which has been in print for approximately 15 years. The goal of the revision was to produce an excellent textbook for second courses in fluid mechanics taken by advanced undergraduate and beginning graduate students while maintaining the book's appeal to instructors who used prior editions. Thus, I sought to maintain or expand the text's fluid mechanics content, while adjusting the text's tone so that this content might be more readily reached by students who may have had only one prior course in fluid mechanics, or who may not specialize in fluid mechanics but do possess appropriate mathematical skills. The entire revision process involved seven steps: (i) formulating a revision plan that was independently reviewed, (ii) agreeing to a formal contract with deadlines, (iii) revising the text, figures, and front matter, (iv) proof reading and correcting copy-edited text, (v) correcting page proofs, (vi) generating the solutions manual, and (vii) tabulating errata. Formulating and executing the

4:40PM L30.00006 Applying the results of education research to help students learn more: an update , RACHEL PEPPER, University of California, Berkeley, C. BAILY, M.D. CABALLERO, S.J. CHASTEEN, B. WILCOX, K.K. PERKINS, S.J. POLLOCK, University of Colorado — Over the past 6 years, the physics faculty at the University of Colorado have worked to transform four core courses in our upper-division undergraduate physics curriculum: Classical Mechanics/Math Methods, Electricity and Magnetism I and II, and Quantum Mechanics. We discuss our transformations as a potential model for transformation of other upper-division courses, such as fluid mechanics. The goal of our transformations was to improve student learning and to develop materials and approaches that other faculty could adopt or adapt. This work began with faculty in the department meeting regularly to define explicit course learning goals, which then served as a foundation for the subsequent course transformations. The development of the curriculum was also guided by the results of observations, interviews, and analysis of student work. We applied the principles of active engagement and learning theory to transform many elements of the course. Reforms included "clicker" questions, tutorials, modified homeworks, and more. In this talk, we will outline the process, the reforms, and present evidence of the effectiveness of these reforms relative to traditional courses. Updates on the progress of ongoing development in Classical Mechanics/Math Methods and second semester Electricity and Magnetism as well as research-based fluid mechanics instructional materials will also be discussed. Our curriculum materials are available at http://www.colorado.edu/sei/departments/physics.htm.

4:53PM L30.00007 An intensive short course in fluid dynamics , G.M. HOMSY, Depts. of Mathematics and of Mechanical Engineering, UBC, BRUCE SUTHERLAND, Depts. of Physics and of Earth and Atmospheric Sciences, Univ. Alberta — This talk reports on an intensive short course in fluid dynamics held recently at the Univ. of Alberta, with an emphasis on the pedagogical objectives and experiences that might be of general interest and utility. The intensive, one week course was intended to introduce students at the undergraduate and beginning graduate level to the topic. No previous exposure to fluid dynamics was assumed. The format consisted of three, 80 minute lectures and one laboratory session per day. Five lecturers (N. Balmforth, M. Flynn, I. Frigaard, G. M. Homsy, B. Sutherland, and R. Sydora) gave 2-3 lectures each, covering a range of topics. Each of three experimental and three computational labs were done by all the students during the week, working in groups of three. This talk will contain details on the lectures, labs, student feedback, and lessons learned, especially with respect to the level of presentations, preparation of labs, and dealing with the diverse backgrounds of the students.

5:06PM L30.00008 Expanding Student International Awareness Through Short-Term Study Abroad Courses With Substantial Engineering Technical Content , FRANK JACOBITZ, THOMAS SCHUBERT, University of San Diego — The efficacy of Compact International Experience (CIE) courses is assessed in this study. These courses were developed with the aim to raise student international awareness while retaining substantial engineering technical content. The courses were motivated by a strong student desire for engineering international studies as well as a drive by the home institution for internationalization of the curriculum. The experiences gained from delivering two distinct three-semester-unit engineering elective courses in three-week time frames in France and Australia are discussed. While the two courses, Topics in Fluid Mechanics and Advanced Electronic Circuit Design, focused on their technical content, the desire for student understanding of the cultural environment and the impact of engineering solutions from a global and societal viewpoint were strong driving factors for each. Assessment validates the hypothesis that CIE courses can successfully deliver substantial engineering technical content while providing an enriching international experience to students.

5:19PM L30.00009 Computational Fluid Dynamics - Applications in Manufacturing Processes , MARIA LAURA BENINATI, AUSTIN KATHOL, CONSTANCE ZIEMIAN, Bucknell University — A new Computational Fluid Dynamics (CFD) exercise has been developed for the undergraduate introductory fluid mechanics course at Bucknell University. The goal is to develop a computational exercise that students complete which links the manufacturing processes course and the concurrent fluid mechanics course in a way that reinforces the concepts in both. In general, CFD is used as a tool to increase student understanding of the fundamentals in a virtual world. A "learning factory," which is currently in development at Bucknell seeks to use the laboratory as a means to link courses that previously seemed to have little correlation at first glance. A large part of the manufacturing processes course is a project using an injection molding machine. The flow of pressurized molten polyurethane into the mold cavity can also be an example of fluid motion (a jet of liquid hitting a plate) that is applied in manufacturing. The students will run a CFD process that captures this flow using their virtual mold created with a graphics package, such as SolidWorks. The laboratory structure is currently being implemented and analyzed as a part of the "learning factory". Lastly, a survey taken before and after the CFD exercise demonstrate a better understanding of both the CFD and manufacturing process.

5:32PM L30.00010 Teaching CFD as a Black Box: A Validation and Verification Approach , JEAN HERTZBERG, University of Colorado, Boulder — There are a number of good reasons for NOT teaching computational fluid dynamics to undergraduates: a reluctance to make room in an already-compressed curriculum, the sophistication of the computational techniques and mathematics involved, the cost of licensing a professional quality code, and above all, the danger that a shallow understanding of CFD will lead to blithely accepted incorrect results. Nevertheless, as today's students enter the workplace they are routinely expected to be able to use CFD and other high level software packages. Industry's response to the necessity of minimally trained engineers using such software is a series of tests prior to accepting the results: verification and validation (V&V), or more specifically independent software verification and validation (ISVV). The verification question asks "is the software producing correct answers, given the inputs?" while the validation question asks "is this the right set of inputs, are the right physics being addressed?" A recent attempt to implement a V&V approach to CFD in the required undergraduate curriculum at the University of Colorado will be described.

Monday, November 19, 2012 3:35PM - 5:45PM –

Session L31 Particle Laden Flows V 33B - Chair: Antonio Ferrante, University of Washington

3:35PM L31.00001 ABSTRACT WITHDRAWN –

3:48PM L31.00002 Turbulent flow computations for high speed shock dominated flows with a one-equation turbulence model , JOHN EKATERINARIS, Prof. Embry-Riddle Aeronautical University — High order accurate discontinuous Galerkin (DG) discretization of the compressible three dimensional Nervier-Stokes equations for hybrid-type meshes is carried out. A general finite element discretization framework is used for all types elements and all computations of the DG method are performed in the regular computational domain for the standard cubic element. Total variation bounded limiters are applied for the standard cubic elements of the computational domain to obtain resolution of three dimensional moving shocks. Three dimensional inviscid flow results for weak and strong shocks over two dimensional configurations showed excellent agreement with other numerical solutions and the experiment. The DG high resolution method is applied for the computation of a moving shock exiting from a cylindrical shock tube and subsequently reflecting from a wall at a distance from the shock tube exit. The Spalart-Almaras one-equation turbulence model is used to obtain turbulent flow solutions for shock dominated flows and near wall turbulence of high speed flow after the exit of the shock. Good agreement with the experiment is found.

4:01PM L31.00003 Radiation-induced turbulence in particle-laden buoyant flows , REMI ZAMANSKY, Stanford University CTR, FILIPPO COLETTI, Stanford University Mechanical Engineering, MARC MASSOT, Stanford University CTR - Ecole Centrale de Paris EM2C, ALI MANI, Stanford University CTR — Particle laden flows are of tremendous importance in oceanography, geophysics, meteorology, astrophysics, and process technology. In the presence of thermal radiation, non-uniformities in particle concentration result in local temperature fluctuations (spatial and temporal), due to the different absorptivity between dispersed and carrier phases. Under the influence of gravity or other acceleration fields, fluid motion is induced by buoyancy, altering the particle distribution and possibly inducing higher non-uniformities. Numerical simulations have been performed which illustrate this effect. It is shown that for a broad range of parameters a (e.g. radiation intensity, acceleration, density ratio and number density) feedback loop between the local particle concentration, the temperature fluctuations and the buoyancy forcing can create and sustain turbulence. Inertial particles are observed to cluster and create high temperature plumes. When the particle response time is comparable to the characteristic lifetime of the plumes, the system exhibits intense fluctuations of turbulent kinetic energy and maximum concentration.

4:14PM L31.00004 Particle-laden turbulence subject to radiation , MOHAMMAD HADI POUR ANSARI, MILAD MORTAZAVI, ALI MANI, Center for Turbulence Research, Stanford — It is well established that particle-laden flows play an important role in numerous technological and natural processes. Although the effect of turbulence on particle concentration is well studied in the literature, little is known about particle-flow systems coupled with radiative heating. Radiation is an active ingredient in many of such systems, including clouds, concentrated solar thermal systems, and astrophysical processes. In these environments the carrier gas is typically transparent and radiation is primarily absorbed by particles. Preferential concentration of particles by turbulence leads to inhomogeneous heating of the mixture over a wide range of length scales. This provides dynamical loops that can alter/force the turbulence. When heating is intense, inhomogeneous expansion of the gas alters the flow. For non-intense heating, the induced buoyancy effect can force the turbulence when a gravitational field is present. We will present results from our calculations, using direct numerical simulation of coupled turbulence-particle transport, demonstrating these effects over a wide range of parameters.

4:27PM L31.00005 Effect of initial cloud shape and orientation on particle dispersion in the accelerated flow behind a shock , SEAN DAVIS, GUSTAAF JACOBS, San Diego State University — We discuss the particle-laden flow development of a cloud of particles with varying initial shapes in the accelerated flow behind a normal moving shock. The effect of initial aspect ratio of a rectangular and elliptical cloud shape as well as the cloud's angle of attack with respect to the carrier flow are considered. Computations are performed with an in-house high order weighted essentially non-oscillatory (WENO-Z) finite difference scheme based Eulerian-Lagrangian solver. Streamlined elliptical shaped clouds produce less particle dispersion in the cross stream as compared to blunt rectangular shaped clouds. Averaged and root mean square statistics of the particle coordinates versus time show that the cloud disperses less with decreasing aspect ratio. The global cloud statistics are comparable for an initially rectangular cloud rotated at 45 degrees as compared to an initially triangular cloud. From observations and statistics we conclude that a particle cloud behaves like a solid body obstruction in the flow at early times, while at later times the particles convect on their inertia.

4:40PM L31.00006 Computational Meso-Scale Study of Representative Unit Cubes for Inert Spheres Subject to Intense Shocks[1] , CAMERON STEWART, Department of Mechanical and Aerospace Engineering, UCSD, FADY NAJJAR, LLNL, D. SCOTT STEWART, JOHN BDZIL, UIUC — Modern-engineered high explosive (HE) materials can consist of a matrix of solid, inert particles embedded into an HE charge. When this charge is detonated, intense shock waves are generated. As these intense shocks interact with the inert particles, large deformations occur in the particles while the incident shock diffracts around the particle interface. We will present results from a series of 3-D DNS of an intense shock interacting with unit-cube configurations of inert particles embedded into nitromethane. The LLNL multi-physics massively parallel hydrodynamics code ALE3D is used to carry out high-resolution (4 million nodes) simulations. Three representative unit-cube configurations are considered: primitive cubic, face-centered and body-centered cubic for two particle material types of varying impedance ratios. Previous work has only looked at in-line particles configurations. We investigate the time evolution of the unit cell configurations, vorticity being generated by the shock interaction, as well as the velocity and acceleration of the particles until they reach the quasi-steady regime.

[1]LLNL-ABS-567694. CSS was supported by a summer internship through the HEDP program at LLNL. FMN's work performed under the auspices of the U.S. Department of Energy by Lawrence Livermore National Laboratory under Contract DE-AC52-07NA27344.

4:53PM L31.00007 Segregation of particles in incompressible random flows:singularities, intermittency and random uncorrelated motion , MICHAEL REEKS, Newcastle University, ELENA MENEGUZ, UK Met Office — We report recent measurements of the segregation of small inertial particles advected via Stokes drag in an isotropic homogeneous incompressible turbulent flow using a full Lagrangian method (FLM) to calculate the compressibility of an elemental volume of particles measured along a particle trajectory. The flow field was generated by a random Fourier mode kinematic simulation (KS) and by DNS. Numerical results show that the average compressibility decreases continuously with time if the value of the Stokes number is below a threshold value 1, indicating that the segregation continues indefinitely. We find that the probability distribution of the compression tends to a Gaussian distribution except in the wings due to the occurrence of singularities in the particle concentration which makes the process highly intermittent. The distribution of singularities over a fixed interval of time for a range of Stoke numbers is shown to be well approximated by a Poisson distribution. Finally, we show that the occurrence of singularities is related to the formation of caustics and the occurrence of random uncorrelated motion (RUM).

5:06PM L31.00008 Heavy particles in compressible homogeneous isotropic turbulence[1] , YANTAO YANG, JIANCHUN WANG, YIPENG SHI, ZUOLI XIAO, State Key Laboratory of Turbulence and Complex System, Center for Applied Physics and Technology and HEDPS, College of Engineering, Peking University, XIANTU HE, Center for Applied Physics and Technology and HEDPS, College of Engineering, Peking University, SHIYI CHEN, State Key Laboratory of Turbulence and Complex System, Center for Applied Physics and Technology and HEDPS, College of Engineering, Peking University — In this talk we study the problem of heavy particles advected by compressible homogeneous isotropic turbulence. We simulate the Eulerian field by a novel WENO-Compact hybrid scheme and track a million heavy particles simultaneously. The heavy particle obeys the dynamic equations $d\mathbf{r}/dt = \mathbf{v}$ and $d\mathbf{v}/dt = -(\mathbf{v} - \mathbf{u})/\tau$, where $\mathbf{r}$ is the location vector, $\mathbf{u}$ and $\mathbf{v}$ are the Eulerian velocity and the particle velocity, respectively. The Stokes number is defined by the relax time τ and Kolmogorov time scale τ_η as $St = \tau/\tau_\eta$. Our study is focused on the effects of the Stokes number on the statistical behaviors of the particles with fixed turbulent Reynolds and Mach numbers. Simulation results reveal that as the Stokes number increases, the intermittency of the particle acceleration weakens. For larger Stokes number, the particle trajectories become smoother, as evidenced by the PDF of trajectory curvature shifting towards smaller value. The particle concentration distribution will also be discussed by investigating the number density and the pair dispersion.

[1]Supported partially by NSFC (Grant No.10921202), the 973 program of China (Grant No. 2009CB724101) and China Postdoctoral Foundation

5:19PM L31.00009 Direct numerical simulation of the erosion of particle beds , ZACHARY BORDEN, University of California, Santa Barbara, LUDOVIC MAURIN, French Air Force Acadamey, ECKART MEIBURG, University of California, Santa Barbara, YULIYA KANARSKA, Lawrence Livermore National Laboratory, MICHAEL GLINSKY, ION Geophysical — Any code that attempts to simulate large scale geophysical flows and their effect on topography needs a way to couple local flow properties to a rate of sediment erosion or deposition. But, the mechanisms responsible for a particle's entrainment from a sediment bed into a flow are poorly understood. To better understand these mechanisms, we employ two- and three-dimensional direct numerical simulations that use a Lagrange multiplier method to enforce solid body motion of, and no-slip boundary conditions on spherical particles within our domain. We apply our code to the configuration of a shear flow over a regularly or randomly packed bed of particles. Results from these simulations will be discussed, and in particular, we focus on the effects of Reynolds number, shear velocity, and initial packing fraction.

5:32PM L31.00010 Particle Interaction in Stratified Fluids[1] , AREZOO ARDEKANI, AMIN DOOSTMOHAMMADI, University of Notre Dame — Hydrodynamics of many industrial and environmental systems is characterized by settling of suspended particles, interacting with each other and the surrounding fluid. Sedimentation of pollutants in the air and settling of marine snow particles in the ocean play an important role in several atmospheric and marine environmental processes. Hydrodynamics of these systems is markedly affected by the presence of vertical density variations that ubiquitously occurs due to temperature or salinity gradients in a fluid column. Despite the widespread implications of settling in stratified media, the fundamental mechanisms of particle interaction are not known to characterize the microstructure of stratified particulate systems. In a homogeneous fluid, a three-stage process, called "drafting, kissing, and tumbling", governs pair interaction of particles settling in tandem and explains the nonlinear behavior of particles in a particulate system. The pressure wake of the leading particle attracts the trailing particle. Upon collision of the particles, an unstable elongated body is formed and tumbles due to inertial effects. In a viscoelastic fluid, the elongated body settling along its long axis is stable and leads to chaining of the particles (drafting, kissing, and chaining). Our recent computational results reveal the role of density stratification on pair particle interaction. We use direct numerical simulations to fully resolve the particle-particle interaction in stratified fluids. The vital role of diffusivity of the stratified agent and the relative importance of inertial and buoyancy forces are investigated.

[1]This work is supported by NSF CBET-1066545.

Monday, November 19, 2012 3:35PM - 5:45PM –

Session L32 Granular Flows III 33C - Chair: Meheboob Alam, JNCASR

3:35PM L32.00001 ABSTRACT WITHDRAWN –

3:48PM L32.00002 Rheology and segregation of a heterogeneous cohesive granular material , PIERRE JOP, CHARLES VOIVRET, EMMANUELLE GOUILLART, Surface du Verre et Interfaces, UMR 125 CNRS/Saint-Gobain, 39 quai Lucien Lefranc, 93300 Aubervilliers — Heterogeneous and cohesive granular flows are investigated by contact dynamic simulations. A given fraction of the spherical grains is cohesive and experience an attractive force at contact with other grains. We study first the rheology of such mixture in a shear plane geometry: we show that a simple law accounts for the influence of the cohesive fraction on the effective friction coefficient and provide some links to the microscopic structure. Then the segregation occurring when flowing under gravity on an inclined plane is investigated. We show that the segregation rate and its intensity depend on the cohesive force and on the amount of cohesive particles.

4:01PM L32.00003 Rearrangements and Rheology in Soft Glassy 2D Material[1] , NATHAN KEIM, PAULO ARRATIA, University of Pennsylvania — We report on simultaneous measurements of shear rheology and microstructure of athermal disordered monolayers of particles ($\sim$6 μm) at an oil-water interface, using an oscillatory magnetic-rod interfacial rheometer. Particle tracking is used to examine the population of dissipative plastic rearrangements that govern mechanical response, and that occur even at small strain. We find that the elastic modulus (G') is approximately 10 times larger than the viscous modulus (G'') at small strains for these soft glassy materials. We describe the statistics of these rearrangements, the growth of viscous dissipation and irreversibility as strain amplitude is increased, and changes in behavior as the system is sheared repeatedly.

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

4:14PM L32.00004 Dynamical Slowing Down for Sheared Granular Materials[1]

, SOMAYEH FARHADI, ROBERT P. BEHRINGER, Duke University — We have performed Couette shear experiments on both circular and elliptical shaped particles below the isotropic jamming point. The dynamics of the system was studied for density regions of $0.85 \leq \phi \leq 0.87$ in ellipses, and $0.80 \leq \phi \leq 0.83$ in disks. In a very small density region, both systems of ellipses and disks evolve slowly in response to continious shear. In particular, we observe that by starting from an essentially unstressed state and applyinmg shear strain, the average displacements of the particles initially grow rapidly, and then slowly decrease for very large strains. In a similar set of experiments performed on disks, slow relaxation was observed as well. However, we observed fundamental differences between the evolution of systems of disks and ellipses. We characterize this slow dynamics by measuring the evolution of velocity profile, density, and orientational order in the course of experiment. Our data suggests that the slow relaxation in ellipses is associated with the small and slow changes in the orientation of particles, which then allow a more efficient packing.

[1]Grant: NSF DMS-0835742, DMR-0906908

4:27PM L32.00005 Gradient and Vorticity Banding Phenomena in a Sheared Granular Fluid

, MEHEBOOB ALAM, PRIYANKA SHUKLA, Jawaharlal Nehru Centre for Advanced Scientific Research, Jakkur PO, Bangalore 560064, India — In many complex fluids, including granular systems, the homogeneous shear flow breaks into alternate regions of low and high shear rates (i.e., shear localization), respectively, when the applied shear rate exceeds a critical shear rate and this is known as gradient banding. On the other hand, if the applied shear stress exceeds a critical value, the homogeneous flow separates into bands of different shear stresses (having the same shear rate) along the vorticity (spanwise) direction, leading to "stress localization." Here we outline a Landau-type nonlinear order-parameter theory for both gradient and vorticity banding phenomena in a sheared granular fluid. Our analysis holds for any general constitutive model, but the specific results will be presented for a kinetic-theory constitutive model that holds for rapid granular flows. Our theory predicts that while the vorticity banding [1] can occur via supercritical/subcritical pitchfork and subcritical Hopf bifurcations in dilute and dense flows, respectively, the gradient banding [2] occurs only via pitchfork bifurcations, both resulting in inhomogeneous states.

[1] P. Shukla and M. Alam, (2012, submitted).
[2] J. Fluid Mech. **666**, 203 (2011); Phys. Rev. Lett. **100**, 068001 (2009).

4:40PM L32.00006 Vibrations and stress relaxation in two-dimensional granular solids

, THIBAULT BERTRAND, CHRISTOPHER MACMINN, COREY S. O'HERN, JOHN WETTLAUFER, Yale University, MARK D. SHATTUCK, City College of New York, ERIC DUFRESNE, Yale University — Wave propagation in granular solids remains poorly understood. The highly non-harmonic vibrational behavior of these systems can lead to events such as large scale rearrangements, fracture, and aging. Extensive studies have been performed to understand the role of non-harmonicity in the melting process and strain weakening of crystalline solids; here, we try to extend these concepts to particulate solids undergoing mechanical excitations that can show stress release through avalanches. We use experiments and numerical simulations to study the onset of non-harmonicity in two-dimensional systems composed of soft particles undergoing uniaxial mechanical excitation. This model system allows us to follow particle trajectories and overall compaction of the packing under uniaxial compression and vibration to disentangle the relation between the loss of harmonicity and the weakening of these 2D granular solids.

4:53PM L32.00007 Reynolds Pressure and Relaxation in a Homogeneous Sheared Granular System[1]

, JIE REN, JOSHUA DIJKSMAN, ROBERT BEHRINGER, Duke University — We describe experiments on the constitutive behavior of a system composed of a disordered collection of frictional disks. We use a novel shear apparatus that avoids the formation of inhomogeneities known as shear bands. We probe the evolution of shear jammed states, occurring for packing fractions $\phi_S \leq \phi \leq \phi_J$, where above ϕ_J there are no stress-free static states, and below ϕ_S, all static states are stress-free. Our linearly sheared, fixed ϕ system exhibits coupling between the shear strain, γ, and the pressure, P, which we characterize by the "Reynolds pressure," and a "Reynolds coefficient," $R(\phi) = (\partial^2 P/\partial\gamma^2)/2$. R depends only on ϕ, and diverges as $R \sim (\phi_c - \phi)^\alpha$, where $\phi_c \simeq \phi_J$, and $\alpha \simeq -3.3$. Moreover, by using asymmetric strain cycles, we find that the observed constitutive relations are limit cycles that are approached logarithmically slowly under cyclic shear. We characterize the relaxation in terms of the pressure asymmetry at cycle n: $\Delta P \simeq -\beta \ln(n/n_0)$. β depends only on the shear cycle amplitude, suggesting an activated process where β plays a temperature-like role.

[1]This work is supported by NSF grant DMR12-06351, NSF grant 0835742, and ARO grant W911NF-11-1-0110.

5:06PM L32.00008 Ratcheting of Granular Polymer with a Spatial Gradient of Excitation[1]

, Y.-C. LIN, Inst. of Physics, Academia Sinica, Taipei, Taiwan (IoP-AS), C.-C. CHANG, IoP-AS, J.-R. HUANG, Nat'l Taiwan Normal University, W.-T. JUAN, J.-C. TSAI, IoP-AS — We study the migration of a short granular chain in response to a sinusoidal vibration whose intensity varies linearly with position. The spatial asymmetry induces ratcheting of the chain that can go, counter-intuitively, either in the direction of lowering the vibration or in favour of its increase, depending upon the position. This spatial divide signals a transition of granular dynamics involving finite-amplitude instabilities. We also demonstrate the roles of cooperative movements both by time-resolving the 3D motion of this macroscopic polymer, and by measuring the persistence and magnitude of its migration that go far above simple predictions based on spatially biased random kicks at the theoretical upper limit.

[1]Supported by IoP-AS and NSC, Taiwan.

5:19PM L32.00009 Jamming of quasi-2D emulsion droplets: Analogies with granular jamming

, ERIC R. WEEKS, KENNETH W. DESMOND, PEARL J. YOUNG, DANDAN CHEN, Physics Dept., Emory University — We experimentally study the jamming of quasi-two-dimensional emulsions. Our experiments consist of oil-in-water emulsion droplets confined between two parallel plates. These are somewhat analogous to granular photoelastic disks, although they are softer and do not experience static friction. From the droplet outlines, we can determine the forces between every droplet pair to within 8% over a wide range of area fractions. Using the data, we observe critical scaling behaviors of the contact numbers and pressure as the jamming transition is approached from above. The scaling behavior agrees well with simulations and is similar to what has been seen previously with photoelastic disks.

5:32PM L32.00010 Patterns, Segregation and Hysteresis in Vertically Vibrated Granular Mixtures

, ISTAFAUL ANSARI, MEHEBOOB ALAM, Jawaharlal Nehru Centre for Advanced Scientific Research, Jakkur PO, Bangalore 560064, India — Granular materials under vertical shaking exhibit a variety of interesting phenomena: undulations, surface instabilities, oscilons, ripples, Leidenfrost state and convection. We have investigated these phenomena by conducting experiments on two types of equimolar *binary* granular mixtures: (i) glass and steel balls both having diameters of $d = 1.0\ mm$ and a density ratio of $\rho_s/\rho_g = 3.0$ and (ii) the delrin and steel balls both having diameters of $1.0\ mm$ and a density ratio of $\rho_s/\rho_d = 5.5$. The particles are held in a quasi-two-dimensional Perspex container which is vibrated harmonically in vertical direction using an electromagnetic shaker. All the experiments are done by increasing the shaking intensity (measured in terms of dimensionless shaking acceleration Γ) while keeping the shaking amplitude A/d fixed. We uncovered many unhitherto reported novel patterns: (i) the Leidenfrost state coexisting with a granular gas, (ii) horizontal segregation within a Leidenfrost state, (iii) granular convection with a floating particle cloud, and (iv) both vertical and/or horizontal segregation with other patterned states. We further show that the transition from the Leidenfrost state to convection in a binary mixture occurs via a hysteretic transition.

Tuesday, November 20, 2012 8:00AM - 10:10AM –

Session M1 Geophysical: Ocean III 22 - Chair: Bruce Sutherland, University of Alberta

8:00AM M1.00001 A model for internal bores in continuously stratified fluids[1] , BRIAN WHITE, University of North Carolina at Chapel Hill — Internal bores, propagating horizontal jumps that connect regions of varying density structure, are generated in the ocean by stratified tidal flow over topography, river plumes, and the breaking of internal waves on a slope. In an undular or soli-bore, internal waves are radiated from the jump and in some cases strong shear and turbulent mixing may occur. Theories for the propagation speed and energy flux through an internal jump have been based on idealized two-layer stratification and require assumptions about the distribution of energy dissipation between internal layers. We discuss a theoretical model for an internal jump in a continuously stratified fluid that applies the Dubreil-Jacotin-Long (DJL) equation (a model for nonlinear solitary waves for which an energy-conserving bore is a well-known limiting solution) and adds a term for depth-dependent dissipation to calculate the velocity and density structure across the jump. The applicability of the continuous model is explored with two- and three-dimensional Navier Stokes calculations of internal bores formed from dam-break initial conditions, focusing on the influence of ambient stratification on energy dissipation and turbulent mixing.

[1]This work was supported by National Science Foundation grant OCE-1029773.

8:13AM M1.00002 Mixing by internal waves impinging on a slope , VAMSI KRISHNA CHALAMALLA, SUTANU SARKAR, Universty of California San Diego — Direct and large eddy simulations are performed to study the mixing that occurs when internal waves interact with critical and near-critical sloping bottom. Different horizontal wave lengths up to $\mathcal{O} \sim 1$ km are considered at a moderate value of Froude number. The pathway from the input wave energy to the irreversible mixing of density field is explored. Diagnostics such as turbulent kinetic energy and density variance budget are used to obtain the phasing of turbulence and associated mixing. Energy transfer to higher harmonics and subharmonics is also quantified.

8:26AM M1.00003 Transport by Internal Waves Near the Boundary of a Lake[1] , CHRIS REHMANN, Iowa State University, DANIELLE WAIN, National University of Ireland at Galway — Because fluxes in stratified water bodies are often controlled by turbulence and mixing at sloping boundaries, determining how the mixed fluid moves from the boundary to the interior is important for estimating basin-wide transport of heat and other scalars. We conducted a tracer-release experiment in a lake that illustrates the importance of advection and dispersion driven by internal waves. Motivated by those observations, we developed an analytical model of transport by internal waves. For tracer releases near the boundary, the velocity field resulting from vertical mode-2 waves consists of oscillatory strain, the oscillating analog of flow near a stagnation point. The horizontal and vertical length scales of the tracer cloud oscillate about the values in the base case with no waves, and the deviation from the base case depends on the ratio of the amplitude of the isotherm displacements and the water depth, as well as the phase at which the tracer was injected. The effect of different modes and the implications for basin-scale transport will also be discussed.

[1]Funding provided by the National Science Foundation through grant OCE-0647253

8:39AM M1.00004 Gravity Currents and Internal Solitary Waves Approaching Slopes , BRUCE SUTHERLAND, DELYLE POLET, University of Alberta, GREG IVEY, University of Western Australia — The theory for the speed of gravity currents and solitary waves usually assumes the depth of the ambient is constant. An adaptation of WKB theory predicts that gravity currents decelerate at a constant rate as they approach as uniform, though small slope. This is bourne out in laboratory experiments of surface gravity currents propagating over a bottom slope even when the rise of the slope is comparable to the head height over its length. However, the deceleration is found to differ by an order of magnitude. The mathematics used to describe internal solitary waves is qualitatively different from that used to describe gravity currents. And yet their dynamics upon approaching a slope is similar. Of particular interest for the off-shore oil industry is where the separation point of the wave trough impacting upon the slope occurs. On the basis of experiments this is predicted as it depends upon the ambient fluid and slope.

8:52AM M1.00005 Three-dimensional structure of vortex shedding beneath internal solitary waves of depression in a two-layered system , PAYAM AGHSAEE, LEON BOEGMAN, Queen's University, Kingston, Canada — Field observations show bed sediment re-suspension occurs in the wake of internal solitary waves (ISWs) of depression traveling over flat and sloping ocean topography. Previous studies suggest that near-bed vortex shedding elevates the near-bed shear and Reynolds stress fields leading to re-suspension. However this work has been limited to investigating the two-dimensional (2D) flow structure and planar PIV results are inconsistent with 2D DNS; vortices ascend higher in the watercolumn in 2D relative to 3D. In this study we present the first three-dimensional (3D) acoustic (ADV) and optical (stereo PIV) observations of vortex shedding beneath ISWs. We show that vortex shedding occurs for smaller values of Reynolds number and pressure gradient parameters in 3D, compared to 2D DNS. The ADV profile data shows transverse velocity fluctuations to be of the same order as horizontal and vertical ones, and this contributes to faster energy dissipation of vortices in 3D relative to 2D simulations, thus limiting the height to which vortices ascend into the watercolumn and potentially transport sediment.

9:05AM M1.00006 Mixing by internal waves , PHILIPPE ODIER, BAPTISTE BOURGET, SYLVAIN JOUBAUD, THIERRY DAUXOIS, Ecole Normale Superieure de Lyon — A key-ingredient in oceanic dynamics is the mixing between waters of different densities. Internal gravity waves, ubiquitous in the ocean, are expected to contribute to these mixing processes. We performed a preliminary experimental study of the mixing induced by a vertical mode-1 or mode-2 internal wave propagating in a linearly stratified fluid. We use a conductivity probe to follow the evolution with time of the density profile, while the wave is continuously generated in the fluid. We observe that a mixing layer is formed at the location of the velocity minima (maximum shear). By measuring the evolution with time of the local Brunt-Väisälä frequency in this region, we are able to derive a mixing velocity. We notice that this mixing velocity increases with the amplitude of the forcing, as expected. We also observe a strong increase of this velocity when the internal wave becomes unstable via a triad resonant interaction (Parametric Subharmonic Instability, see associated abstract by B. Bourget), resulting in the growth of two daughter waves of smaller wave length and frequency.

9:18AM M1.00007 Experimental Study of Parametric Subharmonic Instability in Stratified Fluids , BAPTISTE BOURGET, SYLVAIN JOUBAUD, PHILIPPE ODIER, THIERRY DAUXOIS, ENS de Lyon — Internal waves are believed to be of primary importance as they affect ocean mixing and energy transport. Several processes can lead to the breaking of internal waves and they usually involve non linear interactions between waves. In this work, we study experimentally the Parametric Subharmonic Instability, which provides an efficient mechanism to transfer energy from large to smaller scales. It consists in the destabilization of a primary wave and the spontaneous emission of two secondary waves, of lower frequencies and different wave vectors. We observe that the instability displays a different behavior if the primary wave is a monochromatic vertical mode-1 or a plane wave. Moreover, using a time-frequency analysis, we are able to observe the time evolution of the secondary frequencies. Using a Hilbert transform method we measure the different wave vectors and compare with theoretical predictions. As will be shown further, this instability plays a role in the mixing processes of stratified fluids (see abstract from P. Odier).

9:31AM M1.00008 Reflection of an internal gravity wave beam off a horizontal free-slip surface[1] , QI ZHOU, PETER DIAMESSIS, Cornell University — The reflection of a planar finite-amplitude internal gravity wave beam off a free-slip flat horizontal surface is investigated numerically in a uniformly stratified Boussinesq fluid. Nonlinear effects such as mean currents and harmonics are observed in the wave reflection zone. Mean currents form a stationary, vertically oscillatory, layered structure under the free-slip reflecting surface. The vertical wavelength of the mean-flow layers equals half of the vertical wavelength of the reflecting wave. An empirical predictive model for the steady-state mean flow strength, based on the degree of wave nonlinearity and hydrostaticity, is proposed and subsequently compared to the weakly-nonlinear theory by Tabaei *et al.*, *J. Fluid Mech.*, 2005, *vol.* 526, *pp.* 217-243. Both propagating and evanescent superharmonics are observed, and for waves with steepness of $O(5\%)$, subharmonic instabilities can occur in the late-time of reflection. Other complications to the basic set-up, such as addition of a subsurface mixed layer and spanwise localization of beam, will also be discussed.

[1]Supported by ONR Grant No. N00014-08-1-0235, administered by Dr. Ron Joslin.

9:44AM M1.00009 Internal waves in the Petacalco canyon, Mexico , ANGEL RUIZ-ANGULO, JORGE ZAVALA-HIDALGO, Centro de Ciencias de la Atmosfera, UNAM — On the Mexican coastline, specifically on the Pacific side, there are many submarine canyons. One of the key rolls of coastal submarine canyons is that deep water from adjacent oceanic regions is brought up to the shelf with lots of nutrients enhancing primary production. The head of the Petacalco canyon is located in the Petacalco Bay, in the Pacific Ocean (ca. 17.5N and 102W). During, previous CTD surveys in the area, strong upwelling has been noticed, based on those observations a later survey was designed covering the Petacalco canyon with much larger spatial resolution. Along with those measurements, two thermistor arrays were deployed on the SW crest of the canyon at depths of approximately 60 [m]. The observations, from the thermistor arrays, show large temporal temperature variations with a semi-diurnal frequency. Those variations suggest the presence of internal waves traveling along the canyon axis, if the incidence angle of the internal wave matches the topographic slope results on breaking of internal waves enhancing mixing. This condition occurs at several locations along the canyon axis producing enough mixing of deep oceanic waters with continental waters, increasing the abundance of nutrients in the surrounding region.

9:57AM M1.00010 Settling dynamics of a non-neutrally buoyant particle in stratified fluids , AMIN DOOSTMOHAMMADI, SADEGH DABIRI, AREZOO ARDEKANI, University of Notre Dame — Sedimentation is considered as one of the most important phenomena in characterizing the geochemistry of atmosphere and upper ocean. The vertical variations of temperature and salinity in these environmental systems can have a large impact on settling dynamics of suspended particles. Although the drag increase of the settling particles has been well documented in the recent decade, the fundamental fluid dynamics of unsteady particle-fluid interaction in the presence of density gradients is yet to be explored. Most of the experimental studies have focused on settling in sharp stratified fluids and the numerical works have been limited to steady state flows around axisymmetric neutrally buoyant particles in a linear stratified fluid. We implement a direct numerical simulation of the particle descent in both continuous and sharp stratified fluids to unveil the time dependent response of a non-neutrally buoyant particle. The relative importance of inertia, buoyancy, viscosity and diffusivity is characterized for a wide range of pertinent parameters. Moreover, the quantified investigation of the stratification effects on partial drift volume and time dependent added-mass force will shed light on recent arguments about the importance of drift mechanism in biogenic ocean mixing.

Tuesday, November 20, 2012 8:00AM - 10:10AM –

Session M2 Convection and Buoyancy-Driven Flows VII 23A - Chair: Aline Cotel, University of Michigan

8:00AM M2.00001 Turbulent acidic jets and plumes injected into an alkaline environment , HENDRIK ULPRE, University College London — The characteristics of a strong acidic turbulent jet or plume injected into an alkaline environment comprising of a weak/strong base are examined theoretically and experimentally. A chemistry model is developed to understand how the pH of a fluid parcel of monoprotic acid changes as it is diluted and reacts with the ambient fluid. A standard fluid model, based on a top-hat model for acid concentration and velocity is used to express how the dilution of acid varies with distance from the point of discharge. These models are applied to estimate the point of neutralisation and the travel time with distance within the jet/plume. An experimental study was undertaken to test the theoretical results. These experiments involved injecting jets or vertical plumes of dilute nitric acid into a large tank containing a variety of base salts dissolved in water. The injected fluid contained litmus indicator dye which showed a change in colour from red to blue close to the point of neutralisation. In order to obtain a range of neutralisation distances, additional basic salts were added to the water to increase its pH buffering capacity. The results are applied to discuss the environmental implications of an acidic jet/plume injected into the sea off the South East coast of Great Britain.

8:13AM M2.00002 Convection in a stratified atmosphere: from isolated thermals to the convective boundary layer , CHIEL VAN HEERWAARDEN, JUAN PEDRO MELLADO, Max Planck Institute for Meteorology — We have used direct numerical simulations to study the transition between an isolated turbulent plume penetrating a stably stratified layer and the convective boundary layer, one of the archetypes of the atmospheric boundary layer. Our simulation setup consists of a stably stratified fluid that is heated from below by thermals that form over patches with high surface heat fluxes. The patches are surrounded by regions without surface flux. We have defined a non-dimensional system that allows for studying the transition by varying only one parameter: the ratio of the characteristic length scale of the largest turbulent motions to the distance between two individual patches of high heat flux. By varying its value from zero, which defines the isolated thermal, to the threshold at which the patches are so close that the statistics resemble those of the convective boundary layer, we span the entire transition. Our results show that when the thermals are sufficiently close, the presence of nearby thermals limits their lateral expansion. Rather than merging with neighboring thermals, each thermal organizes into a rising core surrounded by narrow regions of subsiding motions. Nonetheless, in this case many flow statistics resemble those of the convective boundary layer.

8:26AM M2.00003 Turbulent Free Convection Over a Heated Plate , JUAN PEDRO MELLADO, Max Planck Institute for Meteorology — Temporally-evolving turbulent free convection above a heated plate is investigated by means of direct numerical simulations. This study complements previous work in Rayleigh-Bénard convection and in the convective boundary layer. Results show a vertical structure with distinct, overlapping inner and outer layers. Townsend scaling using the surface flux and the molecular diffusivity characterizes the former and Deardorff scaling using the convection scales does so in the latter. It is also observed that the inner layer approaches a quasi-steady behavior. Then, some statistics inside this region coincide with the corresponding ones in classical Rayleigh-Bénard convection if the plate is interpreted as half of the convection cell, which justifies the transfer of results between the two configurations. Despite the moderate Rayleigh numbers, of the order of 10^9, the overlap region between the inner and outer layers starts to show the power-law scaling of the buoyancy variance predicted by the classical similarity theory and seen in the atmosphere. Results also indicate a growth rate of the convective boundary layer in neutral conditions larger than previous estimates derived by extrapolation from related configurations.

8:39AM M2.00004 Numerical simulation of buoyant jet issuing from a seabed , AKIHIKO NAKAYAMA, Construction Engineering Research Institute, HAMID RAHAI, California State University, Long Beach, JEREMY BRICKER, Tokyo Institute of Technplogy — A numerical simulation of effluent mixing in a coastal region has been conducted. The calculation method is a large eddy simulation using the filtered equations of motion and the equations for temperature and salinity without assumption of hydrostatic pressure distribution. The method is tested in a calculation of wall jet and applied to a real flow of effluent discharged from an outfall at the bottom of sea near a coast. The calculation region is about several hundred meters in the horizontal direction and tens of meters in the vertical direction and the time span of about one hour. The results can be fed into larger scale calculation that may not resolve turbulent mixing but calculates dispersion in larger area.

8:52AM M2.00005 Asymmetric character of Rayleigh-Taylor and double-diffusive fingers in reactive systems , L. LEMAIGRE, M.A. BUDRONI, L.A. RIOLFO, Nonlinear Physical Chemistry Unit, Universite Libre de Bruxelles (ULB), Brussels, Belgium, P. GROSFILS, Center for Nonlinear Phenomena and Complex Systems, Universite Libre de Bruxelles (ULB), Brussels, Belgium, A. DE WIT, Nonlinear Physical Chemistry Unit, Universite Libre de Bruxelles (ULB), Brussels, Belgium — Buoyancy-driven flows induced by the hydrodynamic Rayleigh-Taylor or double diffusive instabilities develop symmetrically around the initial contact line when two solutions of given solutes with different density are put in contact in the gravity field. If the solutes affecting the density of these solutions are involved in chemical reactions, changes in composition due to the underlying reaction-diffusion processes can modify the density profile in space and time and affect the hydrodynamic patterns. We show both experimentally and numerically that the resulting chemo-hydrodynamic patterns are asymmetric with regard to the initial contact line as fingers develop here upwards and not both up and downwards as in the non reactive situation. In some cases, the dynamical in-situ generation of the product of the reaction can even lead to the co-existence of different convective modes. The experiments are performed using an aqueous solution of an acid put in contact in a Hele-Shaw cell with a miscible aqueous solution of a base. The numerical simulations are made on the basis of a reaction-diffusion-convection model coupling Darcy's law to the evolution equations of the concentrations of the various chemicals involved. Good agreement between experimental results and theory is obtained.

9:05AM M2.00006 Transport Processes in GaN Deposition in a Chemical Vapor Deposition Reactor , YOGESH JALURIA, JIANDONG MENG, SUN WONG, Rutgers University — A study has been carried out to characterize the metalorganic chemical vapor deposition (MOCVD) growth of Gallium Nitride (GaN). With trimethylgallium (TMGa) and ammonia (NH_3) carried by hydrogen (H_2), as precursors, the entire process which involves fluid flow, heat and mass transfer and chemical kinetics is modeled. A major objective of this work is to examine the dependence of the growth rate of GaN and film uniformity on the flow, as determined by various design parameters and operating conditions involved in the MOCVD process. The results are expected to provide a quantitative basis for the design and optimization of MOCVD system for the fabrication of GaN devices. The study focuses on techniques to guide the impinging flow, and the effect of buoyancy on the resulting flow. Based on the detailed mathematical model and the appropriate chemical mechanisms, a study on the effects of various critical parameters such as the reactor pressure, inlet velocity, susceptor temperature, inflow concentration and rotating speed on the flow and on the growth rate of GaN and thin-film uniformity is conducted for a 3D rotating reactor. The comparison between 3D modeling and previous 2D impinging vertical reactor modeling is presented. The flow and associated transport processes are discussed, indicating approaches to improve the uniformity of the film. In addition, the dependence of the film quality on the inflow profile is also examined, which makes an attempt to minimize the effect of reactor pressure on fluid loss and reduce flow recirculation.

9:18AM M2.00007 Inertial effects in an incompressible stratified Euler fluid in a channel , GIOVANNI ORTENZI, Dipartmento di Matematica e Applicazioni, Universita' di Milano-Bicocca, via Cozzi, 53 - 20125 Milano, Italy, ROBERTO CAMASSA, SHENGQIAN CHEN, Carolina Center for Interdisciplinary Applied Mathematics, Department of Mathematics, University of North Carolina, Chapel Hill, NC 27599, USA, GREGORIO FALQUI, Dipartmento di Matematica e Applicazioni, Universita' di Milano-Bicocca, via Cozzi, 53 - 20125 Milano, Italy, MARCO PEDRONI, Dipartimento di Ingegneria dell'Informazione e Metodi Matematici, Universita' di Bergamo, Viale Marconi 5, 24044 Dalmine (BG), Italy — Inertial properties of an incompressible Euler fluid are discussed in the case of a stably stratified fluid confined between infinite rigid upper and lower horizontal plates in hydrostatic equilibrium at infinity. In this set-up, the possibility of non-conservation of horizontal momentum emerges, despite the fact that only vertical external forces act on the system (an apparent paradox seemingly first noticed by Benjamin, 1986). We show that this phenomenon is a consequence of the rigid lid constraint coupling through incompressibility with the infinite inertia of the far ends of the channel. When inertia is removed by eliminating the stratification or, remarkably, by using the Boussinesq approximation, horizontal momentum conservation is recovered. The pressure imbalance responsible for lack of horizontal momentum conservation and its comparison with direct numerical simulation is explicitly shown for: 1) a two-layer full Euler system with small velocities and density variations, and 2) a long wave one-dimensional model of the same set-up. A side effect detected by the model is the relevance of internal-wave dispersion at short times. Effects on the variation of physical quantities other than momentum, such us the total circulation of the fluid, will be discussed.

9:31AM M2.00008 Characterization of Mixing Between Water and Biofuels[1] , ALINE COTEL, ERICA GREEN, MARINA ACEVEDO, MARGARITA OTERO, AVERY DEMOND, University of Michigan — Currently, gasoline containing ethanol is considered to be among the best alternatives to gasoline. However, the potential environmental impact of a spill of ethanol-based biofuels 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 were observed. An exothermic reaction also occurred between ethanol and water. In the first regime, a turbulent wake is created between the ethanol/glycol and water layers causing the ethanol and glycol solution to entrain and mix into with the water phase. Because the mixed fluid is denser than either parent fluid, a dramatic overturning is possible. The amount of mixing was found to be dependent upon the initial ratio of ethanol to glycol in the parent fluid. 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. The cells at the interface show distinct features of interfacial turbulence, including small transverse waves, denoting that instabilities exist there.

[1]Funding from UM-OVPR and NSF Advance

9:44AM M2.00009 Flux variation and layering in turbulent stratified Taylor-Couette flow , ROSALIND OGLETHORPE, DAMTP, U. of Cambridge, C.P. CAULFIELD, BP Institute & DAMTP, U. of Cambridge, ANDREW W. WOODS, BP Institute, U. of Cambridge — We present new experimental measurements of turbulent mixing in stratified Taylor-Couette flow. We vary the inner radius and rotation rate, and consider both two layer and initially linear stable stratification. With a two layer stratification, we demonstrate that the flux of salt through the interface is a non-monotonic function of a Richardson number $Ri = g\Delta\rho D/(\rho_0 u^2)$, where D and u are characteristic length and velocity scales of the turbulent flow. As predicted by Phillips (1972)DSR **19**, this behaviour implies that a linear stratification will spontaneously form layers of relatively well mixed fluid separated by relatively thin interfaces. We demonstrate the spontaneous development of such layers in experiments where the stratification is initially linear with constant buoyancy frequency N. The depth of these layers $h_L \propto U_H/N$, where U_H is a horizontal velocity scale, and the flux through these layers is independent of h_L. In particular we demonstrate, consistently with Woods $et\ al.$ (2010)JFM **663**, who considered the strongly stratified limit in two layer flow, that the salt flux tends to an asymptotic constant value when the flow is strongly stratified, even when multiple layers are present.

9:57AM M2.00010 Entrainment and mixing dynamics of surface-stress-driven linearly stratified flow in a cylinder , GEORGY MANUCHARYAN, Yale University, C.P. CAULFIELD, BP Institute & DAMTP, University of Cambridge — We consider experimentally a linearly stratified fluid (with constant buoyancy frequency N) in a cylinder of depth H subject to surface stress forcing from a disk spinning at constant angular velocity Ω. A turbulent mixed layer develops bounded by a sharp interface of constant thickness. Its depth $h/H \sim (N/\Omega)^{-2/3}(\Omega t)^{2/9}$. We argue this is a consequence of: the kinetic energy of the mixed layer staying constant with time (as previously observed in a two layer flow by Shravat et al. 2012) the entrainment at the interface being governed entirely by local processes; and the rate of increase of the total potential energy of the fluid being dependent only on the global dissipation rate and the ratio N^2/Ω^2. Below the moving primary interface, we also observe in some circumstances the formation of another partially mixed layer, separated by a secondary interface from the linearly stratified fluid below. Depending on the local flow properties, the secondary interfaces can exhibit rich time-dependent dynamics including drift towards or away from the primary interface, merger and/or decay. The secondary interfaces appear to develop due to the non-monotonic dependence of buoyancy flux on stratification as originally argued by Phillips (1972).

Tuesday, November 20, 2012 8:00AM - 10:10AM –

Session M3 Multiphase Flows: Confined Flows 23B - Chair: Vladimir Ajaev, Southern Methodist University

8:00AM M3.00001 Pressure drop and void fraction during flow boiling in minichannels at different gravity levels , VLADIMIR AJAEV, Southern Methodist University, DAVID BRUTIN, LOUNES TADRIST, Aix-Marseille University (France) — We use mathematical models of two-phase flow to explain recent experimental data on flow boiling in minichannels under the conditions of hypergravity, normal gravity, and microgravity. The experimental data was obtained during parabolic flights and includes simultaneous measurements of void fraction, pressure drop, and heat transfer coefficient. At higher flow rates, void fraction grows linearly along the channel but with different slopes depending on gravity level. Using the models of motion of confined bubbles, we predict the ratio of the slopes of the void fraction profiles which is in excellent agreement with the experimental measurements. At lower flow rates, the void fraction profiles are concave down and eventually flatten away from the channel entrance. This change in the dynamics is explained by a combination of thermal effects in bubble growth, geometric confinement, and bubble coalescense. The relationship between pressure drop measurement results and flow structure is discussed. Finally, experimentally observed heat transfer enhancement under the conditions of microgravity is explained.

8:13AM M3.00002 A mathematically-consistent formulation for evaporation of menisci in microchannels[1] , REZA MONAZAMI, HOSSEIN HAJ-HARIRI, University of Virginia — The problem of evaporation from an extended meniscus enclosed in a rectangular microchannel is investigated. A numerical model is developed to study the effect of channel width, wall superheat, as well as the working fluid. The system of differential equations describing fluid flow, heat transfer and thermodynamics can be reduced to a 4th-order ODE for the thickness of the film from its non-evaporating portion to the base of the meniscus. Prior investigations have used ad-hoc boundary conditions–such as doubling of the thickness–in order to kick start evaporation at some arbitrary point of the non-evaporating film. Such approaches result in severe underprediction of evaporative fluxes. In this talk we present a self-consistent mathematical formulation for the boundary conditions, thereby removing all arbitrariness from the solution process. The results for several channel widths and superheats as well different working fluids indicate that evaporative heat fluxes as high as 10MW/m^2 can be achieved. The results are validated using experiments.

[1]Supported by DARPA.

8:26AM M3.00003 On the stability of two-layer channel flow , AHMED KAFFEL, University of Maryland — The effects of the viscosity, density and surface tension on the hydrodynamic instability of a two-layer viscous stratified shear flows are investigated through a linear stability analysis. In a first stage, we consider the case of isothermal, non-adiabatic, parallel two phase flows. The system of equations for stability are derived and solved numerically using the Chebyshev collocation spectral method. This algorithm is computationally efficient and accurate in reproducing the eigenvalues. The derivation of the asymptotics of these modes shows that our numerical eigenvalues are in agreement with the analytic formula obtained by Yih (1967), Kao and Park (1972), Stergios et al (1988), Pritchard et al (1992) and Pelekasis and Tsamopoulos (2001). These numerical stability results will be used for hydrodynamic problems as a tool to validate the direct numerical solver that solves the coupled two-phase liquid vapor flow dynamics with phase change to characterize the physical mechanisms underlying the quality-heat transfer relationship and thus facilitate the design of microgap channel coolers for specific two-phase heat transfer applications.

8:39AM M3.00004 Experimental Investigation of Fluid and Particle Motion in Shear-Induced Scour[1] , ZHONGFENG AN, PAUL KRUEGER, Department of Mechanical Engineering, Southern Methodist University — A submerged particle bed subjected to fluid shear exhibits particle motion (scour) induced by drag and lift forces from the fluid at sufficiently high shear rates. To investigate this behavior, a particle bed was subjected to fluid shear in a narrow rectangular channel. The flow was driven by a pump for channel Reynolds numbers in the range 3500 – 6000. The particle bed consisted of monodisperse borosilicate glass spheres at several initial particle bed heights. The velocity field of the continuous phase was measured using digital particle image velocimetry (DPIV), while the velocities of the particles were obtained by image segmentation and processing of the dispersed phase from the DPIV images. To aide in visualizing the flow, the working fluid was an aqueous solution of sodium iodide with a refractive index matched to the particles. Comparing the velocity of the two phases, a particle velocity lag was observed at higher elevations, suggesting drag was the dominant fluid force on the particles, while observations of the particle motion indicated that collisions were important near the bed surface. Effects of different flow and initial conditions will be discussed.

[1]This material is based upon work supported by the National Science Foundation under Grant No. 1000908.

8:52AM M3.00005 Simulations of Multiphase Flow in a T-junction and Distributor Header[1] , JEREMY HORWITZ, Member, PURUSHOTAM KUMAR, None, PRATAP VANKA, Member — Multiphase flow is widely encountered in industrial applications including air conditioning and refrigeration systems. In this study, we simulate multiphase flow in complex micro-channels using two approaches: a multiphase Lattice Boltzmann Method (LBM) and a finite volume Volume of Fluid (VOF) method. In LBM, fluids are represented on a mesoscopic scale by particle distribution functions which evolve via a discretized Boltzmann equation. Macroscopic flow variables such as density and velocity are related to the moments of the distribution functions. In contrast, VOF calculates flow variables via three coupled equations: the continuity equation, the Navier-Stokes equation, and the volume-fraction transport equation which tracks the interface between disparate phases. An emphasis is placed on comparison of these schemes to determine their respective advantages in calculation of multiphase flow for these geometries. The principle geometries are a T-junction and multi-branch distributor header. We study bubble-laden flow and immiscible liquid-liquid flow and explore the effect of Reynolds number, buoyancy, and density ratio on the flow physics. Simulation results are compared with experiments.

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

9:05AM M3.00006 Rotation of a spheroidal particle in Couette flow: effects of fluid and particle inertia , TOMAS ROSEN, FREDRIK LUNDELL, MINH DO-QUANG, Linne FLOW Center, KTH Mechanics, Royal Institute of Technology, SE-100 44 Stockholm, Sweden, CYRUS K. AIDUN, G.W. Woodruff School of Mechanical Engineering, Georgia Institute of Technology, Atlanta, Georgia 30332 — Numerical simulations (Lattice Boltzmann simulations with External Boundary Force) of a single prolate spheroidal particle in a Couette flow have been performed, with the aim to study the transitions in particle rotation rate. The system is controlled by two dimensionless parameters, connected to fluid and particle inertia, respectively. Fluid inertia is controlled by the particle Reynolds number, Re_p and particle inertia is controlled by the Stokes number, $St{=}\alpha Re_p$, where α is the density ratio between particle and fluid. Two transitions have been previously reported and are the main focus for this study. The first transition is that with increasing Re_p, a light (buoyant) particle eventually ceases to rotate. The second is that a heavy particle, at a certain St, undergoes a transition from a long period flipping motion to steady rotation with constant angular velocity. The results map out where particle or fluid inertia is more dominant. It was found that multiple solutions exist at constant Re_p, where both periodic rotation and steady state can occur. This transition is determined by a critical density ratio, α_c, for each Re_p and aspect ratio (length/width) of the particle.

9:18AM M3.00007 Discrete Population Balance Multiphase Mixture Modeling of Fragmentation in Fully Developed Turbulent Duct Flow , ADITYA JAYANTHI, JOHN PEDDIESON, Tennessee Technological University — A popular way to model particulate fragmentation is to divide the particle cloud into a finite number of size classes and allow transfer from larger to smaller size classes to simulate fragmentation. To use this approach effectively it is necessary to know the minimum number of size classes required for accurate predictions (size class convergence). In the present paper this question is addressed in the context of fragmentation in a fully developed turbulent duct flow using a multiphase mixture model of the particle cloud. Numerical solutions are obtained for a simple turbulence model and size class convergence is tested using two global measures of particle size distribution for several idealized fragmentation models.

9:31AM M3.00008 On the effect of turbulence on bubbles in a horizontal channel flow , JERRY WESTERWEEL, MARC HARLEMAN, RENE DELFOS, TOM VAN TERWISGA, Delft University of Technology — We present results on the concentration of small gas bubbles in water in a fully-developed horizontal turbulent channel flow. The bubble concentration reaches an equilibrium distribution that is characterized by the Rouse number. Results are obtained by both numerical simulations (DNS) and experiments (PIV) at comparable Reynolds numbers. The gas bubbles in the experiment have a Stokes number that is much smaller than unity, although they do not follow the fluid motion. The volume fraction is sufficiently low to assume one-way coupling. Dispite the low concentration and Stokes number, the bubbles appear to have a preferential concentration at the edge of a downward fluid motion away from the top channel wall. We present a simple model to explain the preferential concentration. For increasing Stokes numbers the rise velocity of individual bubbles in the turbulent channel flow appears to be only 40-50 per cent of the theoretical rise velocity for solid spheres (or gas bubbles in water containing impurities), while the gas bubble Reynolds number remains sufficiently small to assume only linear effects. This is in agreement with earlier reports on the sedimentation of solid particles in a turbulent flow.

9:44AM M3.00009 Linear stability of double-diffusive two-fluid channel flow , KIRTI SAHU, Department of Chemical Engineering, Indian Institute of Technology Hyderabad, Yeddumailaram 502 205, Andhra Pradesh, India, RAMA GOVINDARAJAN, Centre for Interdisciplinary Sciences, Tata Institute of Fundamental Research, Hyderabad, India — Double-diffusive density stratified systems are well studied and shown to display a rich variety of instability behaviour. However double diffusive systems where the inhomogeneities in solute concentration are manifested in terms of stratified viscosity rather than density have been studied far less, and not to our knowledge in high Reynolds number shear flows. In a simple geometry, namely the two-fluid channel flow of such a system, we find a new double-diffusive mode of instability. The instability becomes stronger as the ratio of diffusivities of the two scalars increases, even in a situation where the net Schmidt number decreases. The double-diffusive mode is destabilised when the layer of viscosity stratification overlaps with the critical layer of the perturbation.

9:57AM M3.00010 Experimental investigation on rigid rod-like particles suspensions in archetypal flows by means of Particle Image Velocimetry , ALESSANDRO CAPONE, ALFREDO SOLDATI, University of Udine, GIOVANNI PAOLO ROMANO, University of Rome La Sapienza — Results from an experimental investigation on rigid rod-like fiber particles suspensions in a turbulent pipe jet flow and a backward facing step flow using Particle Image Velocimetry are presented with a focus on turbulence modulation and fiber distribution and orientation. Specific post processing phase discrimination steps allowed simultaneous calculation of carrier and dispersed phase velocities. A turbulent pipe jet flow configuration was investigated in the jet near field region (<5D, where D represents the pipe diameter) within a Reynolds number range 8000-15000, based on D and jet bulk velocity. A turbulent backward facing step flow was analyzed up to 10H downstream of the step at a Reynolds number of 12000, based on step height H and maximum mean velocity. Both flows were tested at two fiber volume concentration C1=0.0001 and C2=0.0005. Results on turbulence modulation are presented as well as fiber concentration and distribution findings. High spatial resolution of backward facing step flow data and specific algorithms allowed fibers orientation detection and orientation distribution calculation. Results from fibers preferential orientation and distribution in the wall-normal direction at selected channel locations and along the flow direction are presented.

Tuesday, November 20, 2012 8:00AM - 10:10AM –

Session M4 Drops X 23C - Chair: Eric Johnsen, University of Michigan

8:00AM M4.00001 ABSTRACT WITHDRAWN –

8:13AM M4.00002 Explosion of Leidenfrost Droplets[1] , FLORIAN MOREAU, University of Liege, PIERRE COLINET, Universite Libre de Bruxelles, STEPHANE DORBOLO, University of Liege — When a drop is released on a plate heated above a given temperature, a thin layer of vapour can isolate the droplet so that it levitates over the plate. This effect was first reported by Leidenfrost in 1756. However, this fascinating subject remains an active field of research in both fundamental and applied researches. In this work, we focus on what happens when surfactant is added to the drop. The aim is to study the influence of a decrease of the surface tension. Surprisingly, as the droplet evaporates, suddenly it explodes. The evolution of the droplet and the resulting explosion are followed using a high speed camera. We show that when a critical concentration of surfactant is reached inside the drop, a shell of surfactant is formed leading to the explosion.

[1]The authors would like to thank FNRS for financial support. This work is financially supported by ODILE project (Contract No. FRFC 2.4623.11)

8:26AM M4.00003 Experimental study of the oscillating interface of a falling drop , SUHWAN CHOI, THOMAS WARD, Iowa State University, Department of Aerospace Engineering — The drop interface oscillation generated from detachment from a nozzle due to gravity are experimentally studied. The fluids used in the experiments are glycerol-water mixtures with viscosities ranging from 0.005 to 0.410 Pa s, mineral oil having a viscosity of 0.0270 Pa s, and DI water with viscosity of 0.0009 Pa s. The drop oscillating is taken by fast camera to make observations. For large drops, where the interface relative to a polar angle may be measured, the periodic deformation is plotted as a function of time. For smaller drops we measure the deformation as switching between an oblate and prolate drop as a function of time. The phenomenon is clearly a function of the fluid viscosity but we seek to propose a pinch-off mechanism for understanding the source of the observed oscillations.

8:39AM M4.00004 ABSTRACT WITHDRAWN –

8:52AM M4.00005 Interaction of droplets in recirculation regions within microfluidic systems , NASTARAN GHAZI, Columbia University, ASHKAN HOSSEINI, SHAHAB SHOJAEI-ZADEH, Rutgers, The State University of New Jersey — We investigate the interaction of oil droplets in continuous water phase as they travel across the streamlines of a recirculation region using microfluidic devices. Oil droplets are first generated using hydrodynamic focusing and then enter a recirculation region. The droplets then keep recirculating until they are pushed out by the incoming ones. We show that the frequency of droplet generation, viscosity contrast (oil to water), and geometry determine which droplets to stay in the recirculation region and which one to leave. Using flow field simulations, we investigate the migration of droplets and their trajectories based on the geometry of the recirculation region, the bubble size, and fluid properties. Under favorable conditions, when droplets interact within the recirculation region for long enough time, the film thickness that separates the two interfaces reduces and droplets will coalesce. The proposed design thus provides a suitable platform to study droplet coalescence within microfluidic devices.

9:05AM M4.00006 Experimental Investigation of Gravity- and Wind-Forced Drop Stability[1] , JASON SCHMUCKER, EDWARD WHITE, Texas A&M University — The stability of drops on surfaces subject to forcing by wind and gravity is relevant to heat exchangers, fuel cells, and aircraft icing. To investigate this phenomenon, drops were placed on the rough aluminum floor of a tiltable wind tunnel and brought to critical conditions over a range of drop size, inclination angle, and flow speed. A technique capable of measuring full 3D drop profiles was used to investigate the drops' evolution toward runback. The measurement uses a comparison of the surface speckle pattern captured in an overhead drop image with a corresponding image of the dry surface to measure drop shape. Drops forced by airflow alone are found to shed at a Weber number of 8.0 ± 0.5 for this system with advancing and receding contact angles of $\theta_a = 63.5^\circ \pm 3.7^\circ$ and $\theta_r = 8.2^\circ \pm 1.5^\circ$. Drops at larger surface inclinations shed at lower Weber number. From reconstructed drop profile sequences, the evolution of contact lines, drop profiles, and contact angle distributions are detailed. Contact line adhesion forces are integrated and related to the forcing air velocity. Drops whose stability limits are dominated by gravity are found to exhibit significantly different evolution toward runback than those dominated by airflow.

[1]Supported by NSF Grant CBET-0828469.

9:18AM M4.00007 Physics of drop formation at a step , ZHENZHEN LI, SAMUEL METAIS, Microfluidics MEMS and Nanostructures Laboratory -ESPCI, ALEX LESHANSKY, LEN PISMEN, Technion-Israel Institute of Technology, Israel, PATRICK TABELING, Microfluidics MEMS and Nanostructures Laboratory -ESPCI, MICROFLUIDICS MEMS AND NANOSTRUCTURES LABORATORY -ESPCI COLLABORATION, DEPARTMENT OF CHEMICAL ENGINEERING IIT ISRAEL COLLABORATION — When a confined liquid stream surrounded by another immiscible liquid flows from a shallow channel into a deep reservoir, droplets can be generated at the entrance of the reservoir. We perform series of experiments in this geometry in a microfluidic system and compare our observations to a recent theory developed by A. Leshansky and L. Pismen. We observe that below a critical capillary number, Ca, small monodispersed droplets are formed. We give scaling explanation on their sizes. Above the critical Ca, the stream takes a quasi-steady shape feeding a large droplet formed in the reservoir. Theoretical arguments are presented explaining the transition between the two regimes and the shape of the interface, that separates the two liquids. The experiments confirm the theory at a quantitative level.

9:31AM M4.00008 Janus droplet motion in an external flow , SERGEY SHKLYAEV, Department of Chemical Engineering, University of Puerto Rico, ANDREY IVANTSOV, Institute of Continuous Media Mechanics UB RAS, MISAEL DIAZ, UBALDO M. CORDOVA-FIGUEROA, Department of Chemical Engineering, University of Puerto Rico — We consider a hydrodynamics of a Janus droplet, which consists of two hemispherical domains occupied by different liquids. The simplest problem, a Janus droplet in a uniform at infinity flow, is analyzed. The interfaces are assumed weakly deformable. It is shown, that the velocity field can be represented as a superposition of two fields: for internal surface (i) normal and (ii) parallel to the external flow. In case (i) the flow is axisymmetric; the force imposed on the droplet is found by summation of the series. It is worth noting, that even for equal internal viscosities, the solution for a simple drop [1,2] is not reproduced. Indeed, the internal impermeable interface prohibits a flow of Hadamard-Rybczynski type. Weak deformation of the interfaces is found; it is shown that deformation of the internal surface is larger than that of the drop surface. In case (ii) expansion in Lamb's functions is applied; both the torque and force are found. It is also shown that stable configuration of a torque-free droplet corresponds to case (i) with less viscous fluid on the upstream face.

[1] J. S. Hadamard, Compt. Rend. Acad. Sci. 152, 1735 (1911).
[2] W. Rybczynski, Bull. Acad. Sci. Cracovie (ser. A), 40 (1911).

9:44AM M4.00009 ABSTRACT WITHDRAWN –

9:57AM M4.00010 Air Entrapment for Liquid Drops Impacting a Solid Substrate[1] , YUAN LIU, PENG TAN, LEI XU, The Chinese University of Hong Kong — Using high-speed photography coupled with optical interference, we experimentally study the air entrapment during a liquid drop impacting a solid substrate. We observe the formation of a compressed air film before the liquid touches the substrate, with internal pressure considerably higher than the atmospheric value. The degree of compression highly depends on the impact velocity, as explained by balancing the liquid deceleration with the large pressure of compressed air. After contact, the air film expands vertically at the edge, reducing its pressure within a few tens of microseconds and producing a thick rim on the perimeter. This thick-rimmed air film subsequently contracts into an air bubble, governed by the complex interaction between surface tension, inertia and viscous drag. Such a process is universally observed for impacts above a few centimeters high.

[1]Hong Kong GRF grant CUHK404211 and direct grant 2060418.

Tuesday, November 20, 2012 8:00AM - 10:10AM –

Session M5 Computational Fluid Dynamics VII 24A - Chair: Tim Colonius, California Institute of Technology

8:00AM M5.00001 A numerical method for Stokes flow in a complex geometry coupled to dynamic rigid structures and filaments , TAMAR SHINAR, University of California, Riverside, MICHAEL SHELLEY, New York University — We present a numerical method for the simulation of Stokes flow coupled to fixed and dynamic rigid bodies. The method uses an immersed boundary formulation for the fluid problem, where the problem domain is embedded in a periodic domain, and the boundary conditions are enforced through singular source terms. Rigid body generalized coordinates and velocities are used for the structures, though the method could be extended to deformable structures as well. The structure forces are nonlinear in general and we solve the coupled problem using a Newton-Krylov method, where the associated linear systems are symmetric indefinite. The coupling forces between the fluid and structures are treated in a fully implicit manner, making the choice of stable time step independent of those forces. We demonstrate the method by studying the dynamics of mitotic spindle positioning in a model of a single-celled *C. elegans* embryo.

8:13AM M5.00002 Coupled level-set CURVIB method for fluid-structure interaction simulations of arbitrarily complex floating rigid bodies[1] , ANTONI CALDERER, SEOKKOO KANG, FOTIS SOTIROPOULOS, St. Anthony Falls Laboratory, University of Minnesota — We develop a fluid-structure interaction (FSI) model for simulating arbitrarily complex floating rigid bodies interacting with nonlinear free-surface flows. The FSI curvilinear immersed boundary (CURVIB) method of Borazjani et al. (JCP 2008) is integrated with the LES CURVIB method of Kang et al. (Adv. in Water Resources 2010) and the recently developed level set-CURVIB method (Kang and Sotiropoulos, Adv. in Water Res. 2012) to develop a powerful method for simulating 3D nonlinear turbulent free-surface flows. To demonstrate the predictive capabilities of the method and its ability to simulate non-linear free-surface phenomena, such as breaking waves, we apply it to simulate various cases involving 2D/3D free surface-rigid body interactions. The computed results are shown to be in excellent agreement with available experimental measurements.

[1]This work was supported by the US Department of Energy grant DE-EE0005482 and the Minnesota Supercomputing Institute

8:26AM M5.00003 The immersed interface method without interface parametrization , GLEN PEARSON, SHENG XU, Southern Methodist University — To simulate fluid-solid interaction or two-fluid flows on Cartesian grids by the immersed interface method, we incorporate into a numerical scheme the jump conditions of the first- and second-order Cartesian derivatives of the velocity and pressure. These Cartesian jump conditions can be systematically derived from the principal jump conditions for the velocity and the pressure [Sheng Xu, Z. Jane Wang, Systematic derivation of jump conditions for the immersed interface method in three dimensional flow simulation, SIAM J. Sci. Comput. Vol 27, No. 6, pp. 1948-1980.], i.e. the jump conditions of the velocity and the pressure, their normal derivatives and their Laplacians. However, this previous derivation requires the global parametrization of a fluid-solid or two-fluid interface. In this talk, we present a new derivation which is based on the triangulation of an interface and avoids the interface parametrization. The new derivation makes the immersed interface method more robust for applications. We will test our new derivation by solving Poisson equations with discontinuous solutions across triangulated interfaces.

8:39AM M5.00004 Fully resolved immersed electrohydrodynamics for target-detection, particle motion, and self propulsion[1] , 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, NEELESH A. PATANKAR, Department of Mechanical Engineering, Northwestern University — Motion of particles, rigid or deforming, through conductive fluid media under the presence of electric fields require the solution of coupled electrodynamics and hydrodynamics equations. In this work we present a numerical method for modeling such coupled equations in an adaptive mesh refinement and immersed body framework. The methodology permits us to locally resolve high electric field gradients and boundary layers near the fluid-structure interfaces at a moderate computational expense. Using such a framework a broad range of problems such as "electrolocation" (a technique used by knifefish to detect its target due to the distortion of self generated electric field by a prey in its vicinity), dielectrophoretic motion of particles in microfluidic channels, development of artificial "electrosense" for underwater vehicles, among others, can be addressed.

[1]NSF support is gratefully acknowledged.

8:52AM M5.00005 A local grid refinement curvilinear immersed boundary method for multi-resolution simulations of complex turbulent flows[1] , DIONYSIOS ANGELIDIS, FOTIS SOTIROPOULOS, St. Anthony Falls Laboratory, Department of Civil Engineering, 2 Third Avenue SE, Minneapolis, MN 55414, USA — A novel numerical method for multi-resolution real life simulations is developed, solving the Navier-Stokes equations on curvilinear locally refined grids in conjunction with the Curvilinear Immersed Boundary Method (CURVIB) (Ge and Sotiropoulos, J. Comp. Phys. 2007). The governing equations are discretised on 3D unstructured grids using a hybrid staggered/non-staggered approach and employing the fractional step method; thus discretising the momentum equations fully implicitly using second-order scheme in time. The momentum equations are solved with the matrix free Newton-Krylov method, while the Poisson equation is solved by implementing the algebraic multigrid method (AMR). By using the unstructured approach, the memory requirements are minimized since all the data are stored in one dimensional arrays. Thus, adaptive cells' splitting and merging is related with memory allocation and deallocation respectively. Partitioning of the unstructured grid and parallel computing enhance the solver's performance, making the code a powerful tool capable for multi-resolution calculations of complex turbulent flows involving a large disparity of spatial scales, such as flows past wind turbine arrays. The flow solver is developed to enable large-eddy simulations, with low computational cost.

[1]This work was supported by Department of Energy DOE (DE-EE0005482).

9:05AM M5.00006 A Numerical study of ablative flow driven by thermodynamics and kinetics[1] , RYAN CROCKER, YVES DUBIEF, University of Vermont, CHRISTOPHER WHITE, University of New Hampshire — The main focus of this research is to elucidate the relationship between ablative/erosive flows and their interaction with a spatially dynamic boundary conditions. The fluid-solid interface is described by a level-set (LS) approach, enabling efficient and accurate computation of wall-normal vectors and other geometrical properties. Boundary conditions at the interface are imposed using immersed boundary methods (IBM). Our LS/IBM ablation algorithm is able to simulate various ablation processes, in particular ablation by chemical reactions and phase-change. The momentum boundary conditions are handled through cut cell IBM applied implicitly. This method is fully mass conserving and eliminates issues with spurious pressure oscillations. Scalar boundary conditions are simulated with a form of the ghost fluid method, with known thermal and chemical physics, to interpolate values across the fluid/solid interface after they are decoupled. We will discuss two applications of this algorithm. The first is the simulation of an experiment of the oxidation of a carbon material at around 1000K conducted at NASA AMES. The second focuses on the interactions between heated wall-turbulence and a low-melting point material in collaboration with parallel experiments at University of New Hampshire.

[1]We gratefully acknowledge the financial support of NSF, grand No. CBET 0967857, and NASA, grant No. NNX11AM07A, and the computational support of the Vermont Advanced Computing Center at the University of Vermont.

9:18AM M5.00007 An Immersed Boundary Method for the Incompressible Navier-Stokes Equations Based on the Lattice Green's Function Method[1] , SEBASTIAN LISKA, TIM COLONIUS, California Institute of Technology — A parallel, three-dimensional immersed boundary method is developed to solve external, viscous incompressible flows on an infinite domain. The equations are formally discretized on an infinite staggered Cartesian grid. An advantage of the infinite grid is automatic commutativity of operators and associated conservation properties. The Lattice Green's Function (LGF) method is used to reduce the solution to a finite portion of the grid. The LGF method uses the fundamental solutions of discrete operators on infinite grids analogously to solving continuous inhomogeneous differential equations using Green's functions. The differential-algebraic-equations that describe the temporal evolution of the discrete momentum equation, the incompressibility constraint, and the no-slip constraint are numerically solved by combining an integrating factor technique for the viscous term and a half-explicit Runge-Kutta scheme for the convective term. A nested projection technique is used to efficiently solve the algebraic system of equations involved in each stage of the time march. Fast solutions to the discrete elliptic problems that arise from the projection technique are obtained through a new solver based on the LGF that shares constructs from fast multipole methods and tree-algorithms. Results for three-dimensional test problems are presented, and the performance and scaling of the present implementation are discussed.

[1]Support of the AFOSR through grant FA9550-09-1-0189 is gratefully acknowledged.

9:31AM M5.00008 Multi-Implicit Blob Projection Method , LAUREN FOVARGUE, Rensselaer Polytechnic Institute — Many problems in biological fluid-structure interaction have been studied with C. S. Peskin's immersed boundary method (IBM). This method defines a relatively simple mathematical modeling framework, and allows for the use of standard fluid solvers. Often, when this is evaluated computationally in many applications, a very small time step is required. This time step is not restricted by accuracy, but stability, causing the temporal problem to be stiff. Previous attempts to address the stability restriction of IBM use semi or fully implicit schemes that require the code, including the fluid solver, to be rewritten, and these have yet to be implemented in application focused studies. Here, new ideas for addressing the computational cost of IBM are presented, which rely on a novel method for splitting the spatial field into stiff and non-stiff components. With this splitting, the impact on the largest stable time step and computational cost for stiff problems is investigated through a multi-implicit technique, which focuses on the treatment of the immersed boundary, where no changes to the fluid solver are necessary.

9:44AM M5.00009 A sharp, robust, conservative cut-cell immersed boundary technique , PETER BRADY, OLIVIER DESJARDINS, Cornell University — Simulation of solid-fluid systems with complex boundaries can be greatly simplified using immersed boundary (IB) methods. IB methods have have been used for many years because they provide an alternative to using a full body-fitted mesh, which often requires an unstructured CFD code. 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. Yet, discrete conservation can be obtained by using a cut-cell IB approach, where the cells that intersect with the solid body are cut such that they become body-fitted. Challenges typically associated with cut-cell methods include: expensive geometry manipulations (especially in three dimensions), the creation of arbitrarily small cut-cells and a modified discretization in those cut-cells. We address these issues by representing the interface implicitly and computing the cut-cell geometry using a marching tetrahedra algorithm. The discretization is then modified in the cut-cells using this geometric information. The code is verified and several techniques for handling small cells and the application of boundary conditions at the IB surface are evaluated using the method of manufactured solutions.

9:57AM M5.00010 An improved direct-forcing immersed boundary method for fluid-structure interaction of a flexible filament[1] , XING ZHANG, XIAOJUE ZHU, Institute of Mechanics, Chinese Academy of Sciences — We present an improved immersed boundary method for the simulation of fluid structure interaction (FSI) of a slender body. Our numerical method is based on the one proposed by Wang and Zhang (J. Comput. Phys. 30:3479-3499, 2011). Although an accurate prediction of total force can be achieved by using this method, unphysical spatial oscillation is observed in the force distribution. This oscillation is detrimental to the prediction of structure response in FSI. In this work, several modifications are made to improve this method. Firstly, the implicit forcing is replaced by an explicit forcing. Secondly, a more consistent way of computing each component of the forcing on a staggered mesh is proposed. Thirdly, for a slender body of zero thickness, the discrete delta-function with a "negative-tail" is adopted for the interpolation at the endpoints. Numerical simulations are performed to test the efficacy of the modifications. It is found that the measures taken successfully reduce the oscillation and the results obtained agree well with those from the literatures.

[1]This work was supported by NSFC 10872201.

Tuesday, November 20, 2012 8:00AM - 9:57AM –

Session M6 Microfluidics: Capillary II 24B - Chair: Thomas Cubaud, Stony Brook University

8:00AM M6.00001 Dynamics of a small number of droplets moving in microfluidic Hele-Shaw cells , BINGQING SHEN, MARIE LEMAN, PATRICK TABELING, MATHILDE REYSSAT, ESPCI — We investigate both theoretically and experimentally the dynamics of a small number of droplets moving in microfluidic Hele-Shaw cells. We derive simplified equations in the limit of large friction against the wall. These equations provide a quantitative description of the dynamics observed in the experiments. Similar conclusions hold for the influence of the walls.

8:13AM M6.00002 Multiple bubble propagation modes in Hele-Shaw cells of variable depth , ALICE THOMPSON, ANNE JUEL, ANDREW HAZEL, University of Manchester — Experimental investigations of long air bubbles displacing viscous oil in axially uniform rectangular channels have shown that symmetric partial occlusion of the cross-section can induce a variety of bubble morphologies, which may find practical application in microdevices. For sufficiently high steady flow rates, the bubbles switch from centred to asymmetric configurations,[1] and spatial oscillations can develop behind the tip in intermediate regimes.[2] We show that all observed morphologies are reproduced in a two-dimensional depth-averaged model, similar to that describing Saffman–Taylor fingering, but with a spatially varying channel height. The resulting equations are solved numerically, using the finite-element library `oomph-lib`[3] and we present the results of a bifurcation analysis that complements the existing experimental data. The qualitative agreement between 2D model and 3D experiments indicates that the complex behaviour arises solely from the enforced change in transverse curvature of the air-oil interface.

[1]de Lózar et al., *Phys. Fluids* **21**, 101702, (2009)
[2]Pailha et al., *Phys. Fluids* **24**, 021702, (2012)
[3]Heil & Hazel, *Lect. Notes Comput. Sci. Eng.* **53**, (2006)

8:26AM M6.00003 Modeling sessile droplets on hydrophobic surfaces with spatially varying contact angle , OISTEIN WIND-WILLASSEN, MADS PETER SORENSEN, Technical University of Denmark — We present a mathematical model that we have developed in order to numerically investigate droplets deposited on hydrophobic surfaces with spatially varying contact angle, $\theta(\mathbf{r})$. If a gradient in θ is induced on the surface it is possible to guide the flow. The model solves the Navier-Stokes equation on a time-dependent domain $\Omega(t)$ by the use of the Finite Element Method with a moving mesh (Arbitrary Lagrangian-Eulerian, ALE). A Navier slip boundary condition at the fluid-solid interface has been implemented, and at the free surface the Young-Laplace equation is used. Results for 1) drops deposited on an inclined plane will be presented, along with 2) results for a drop oscillating in a "potential well" made from rapidly (but smoothly) changing the contact angle. In case 1) we examined the internal flow field of the drop and, according to the model, a rotating flow builds up as the overall velocity increases. In case 2) the drop oscillates in a damped way since viscous friction damps out the energy. There seem to be a non-linearity between the strength of the potential and the amplitude, decay length, and frequency of the oscillations. We suspect this is due to some preferred internal oscillatory state/eigenmode of the drop.

8:39AM M6.00004 Modification of the effective contact angle by means of particle collection and the generation of armored bubbles , FARZAM ZOUESHTIAGH, MICHAËL BAUDOIN, University of Lille 1, Institut d'Electronique de Microélectronique et de Nanotechnologie (IEMN) CNRS 8520 — We report on an experimental work where hydrophobic particles are collected by a moving water-air meniscus in a capillary tube. The results show that under certain conditions the collected particles can position themselves on a granular monolayer at the liquid-air interface and move with it. A continuation in the particle collection by the meniscus results in the decrease of the effective liquid-water contact angle (CA) in the capillary tube. At zero CA an "air finger" with its walls covered by particles forms. The finger displays stick-slip behavior in its motion due to solid friction of the particles with the wall. This friction eventually provokes its pinch-off which results in the formation of a cylindrical armoured bubbles.

8:52AM M6.00005 Capillary Rise in Tubes with Interior Corners[1] , MARK WEISLOGEL, TYLER MILHEM, BRIAND OAKS, Portland State University — The classic problem of sudden capillary rise in tubes (i.e. channels) with one or more interior corner is revisited from the perspective of a first tier advancing bulk meniscus of constant curvature that feeds second tier advancing corner flows. Asymptotic solutions are presented for the visco-capillary limit that extends the Lucas-Washburn solution to such geometries. These "compound capillary" flows are common but overlooked due to the relatively small impact on overall flow rate for typical geometries such as the square tube. However, for more acute sections, the second tier corner flows are significant and cannot be ignored. Such geometries may be exploited for a variety of applications including passive phase separations. Supportive experimental data are presented using both micro-scale terrestrial and macro-scale drop tower tests.

[1]NASA NNX09AP66A: GRC, NASA NNX10AK68H: Oregon Space Grant Consortium

9:05AM M6.00006 Thermocapillary Levitation of Nanoliter-Volume Droplets and Extension to Two-Phase Systems[1] , JAMES BLACK, G. PAUL NEITZEL, Georgia Institute of Technology — The development of a novel method of droplet levitation to be employed in lab-on-a-chip (LOC) applications relies upon the mechanism of thermocapillary convection (due to the temperature dependence of surface tension) to drive a layer of lubricating gas between droplet and substrate. The fact that most droplets of interest in LOC applications are aqueous in nature, coupled with the fact that success in effecting thermocapillary transport in aqueous solutions has been limited, has led to the development of a technique for the controlled encapsulation of nanoliter-volume water droplets within a shell of inert silicone oil. Previously, microliter-volume single-phase silicone-oil droplets have been levitated. This work aims to extend this technique to nanoliter-volume single- and compound-phase oil and water droplets as well as ascertain how the fluid-fluid interface affects the internal convective currents driven by the surface flow in compound-phase systems.

[1]Supported by NASA and NSF

9:18AM M6.00007 Microfluidic dissolution of CO_2 bubbles in viscous oils[1] , MARTIN SAUZADE, THOMAS CUBAUD, Stony Brook University — We experimentally study the interrelation between the dissolution of carbon dioxide bubbles and microfluidic multiphase flows. Individual bubbles are generated in silicone oils at the junction of a hydrodynamic focusing section. High-speed imaging is used to track individual bubbles and monitor their shape and velocity as they experience a reduction in size due to gas diffusion. The early diffusive behavior is analyzed using computational routines developed to reconstruct the evolving 3D volume of elongated bubbles from their 2D contour. In particular, we examine the fast initial diffusive behavior, which is characterized as a function of the oil molecular weight - from low to high viscosity - gas inlet pressure, flow rates, and microgeometries.

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

9:31AM M6.00008 Phase-locked confocal micro-PIV measurement of three dimensional flow structure of transient droplet formation mechanism in T-shaped micro junction , MASAMICHI OISHI, HARUYUKI KINOSHITA, TERUO FUJII, Institute of Industrial Science, The University of Tokyo, MARIE OSHIMA, Interfaculty Initiative in Information Studies, The University of Tokyo — This paper aims to investigate a mechanism of microdroplet formation at a micro T-shaped junction using a "phase-locked multicolor confocal micro-PIV (Particle Image Velocimetry)" technique. The multicolor system can measure dynamic behavior of each phase of multiphase flow separately and simultaneously. The phase-locking technique is necessary to reconstruct three-dimensional flow field from two-dimensional confocal micro-PIV measurement data. We successfully obtained each temporal phase of periodic phenomenon non-invasively by detecting passage of droplet from transmitted light of optical proximity sensor. Additionally, the phase-locking technique also enables picking up same condition of formation frequency, size and translational velocity of droplet to minimize instability of droplet formation phenomenon. As a result, three-dimensional flow structure of the droplet formation was successfully reconstructed and the droplet formation mechanism was investigated by the flow interaction between each phase.

9:44AM M6.00009 Formation of partially wetting droplets in square microchannels[1] , BIBIN M. JOSE, THOMAS CUBAUD, Stony Brook University — We experimentally study the formation and evolution of partially wetting droplets in microchannels made of glass and silicon. Droplets are steadily generated by focusing water in an external phase of silicone oil using square channels. To probe the influence of the capillary number on droplet and wetting dynamics in confined geometries, the oil viscosity is varied over four decades. At low capillary numbers, we observe the formation of contact lines and the nucleation of dewetting patches in the thin film surrounding elongated droplets. By contrast, small and lubricated droplets are produced at large capillary numbers. The dynamic wetting properties of microfluidic segmented flows are compared with measurements performed using a contact angle goniometer.

[1]This work is supported by NSF (CBET-0932925)

Tuesday, November 20, 2012 8:00AM - 9:44AM –

Session M7 Geophysical: General V 24C - Chair: Hieu Pham, University of California, San Diego

8:00AM M7.00001 Buoyancy effects in the spatially-evolving wake of a sphere at Re=3,700 , MATTHEW DE STADLER, SUTANU SARKAR, University of California San Diego — Direct numerical simulation is used to simulate spatially-evolving flow past a sphere in a stratified fluid. A Cartesian grid is used along with an immersed boundary method to represent the sphere inside the domain. The Reynolds number of 3,700 is chosen so that the wake behind the sphere is turbulent. The emphasis of the present study is on the near to intermediate wake as buoyancy effects become dominant. A comparison is made between an unstratified wake and a wake at a Froude number of 3. Statistics of interest include the defect velocity, wake dimensions, turbulence intensities, mean kinetic energy, turbulent kinetic energy and associated budgets. Visualizations of the vortical structures in the wake and the internal wave field will also be provided and discussed.

8:13AM M7.00002 The Lagrangian energetics of stably stratified turbulence in the Boussinesq approximation with non-linear equation of state , SEUNGBUM JO, KEIKO NOMURA, JAMES ROTTMAN, University of California, San Diego — There has been a recent resurgence of interest in determining the consistency of the Boussinesq approximation to describe the coupling of the dynamics and thermodynamics of turbulent stratified flows. In particular, there is some debate over how energy is converted from internal to mechanical energy in this approximation. Moreover, the effect of the non-linear equation of state and the strength of stratification on the internal energy is still unclear. To gain some insight into these issues, we derive the evolution equations of the different forms of energy for Boussinesq stratified flows with variable volumetric expansion coefficient in the Lagrangian frame. This analysis allows us better physical insight into these issues and allows us to show explicitly how energy is converted between internal and mechanical energy and how significant the internal energy is under these conditions. The physical significance of these results will be discussed.

8:26AM M7.00003 A zero-equation closure model for wall-bounded stably stratified flows[1] , FARID KARIMPOUR[2], SUBHAS KARAN VENAYAGAMOORTHY[3], Colorado State University — In this study, we propose a parameterization for the turbulent Prandtl number (Pr_t) for stably stratified wall-bounded flows. To date, most of the widely used parameterizations for Pr_t for stably stratified flows are based on data from homogeneous flows and are usually formulated as functions of the gradient Richardson number (Ri_g). The effect of the wall boundary is completely neglected. We introduce a modified parameterization for Pr_t that takes into account the inhomogeneity caused by the wall coupled with the effects of density stratification. We show that in wall-bounded flows, the turbulent Prandtl number has a different behavior from homogeneous flows. We evaluate the new parameterization by using a zero-equation turbulence model for the eddy viscosity that was proposed by Munk and Anderson in 1948 to simulate a one-dimensional stably stratified channel flow. Comparison of the one-dimensional simulation results with direct numerical simulation of stably stratified channel flow results show remarkable agreement. We also compare other commonly used parameterizations of Pr_t for homogeneous flows to highlight their shortcomings in predicting both momentum and scalar mixing correctly in wall-bounded flows.

[1]Funded by the National Science Foundation
[2]Graduate Student
[3]Assistant Professor

8:39AM M7.00004 Pathways to dissipation in strongly rotating and stratified turbulent systems[1] , ENRICO DEUSEBIO, ERIK LINDBORG, Linné FLOW Centre, Royal Institute of Technology, KTH — Geophysical flows are strongly affected by rotation and stratification at very large scales ($\approx 10^3$ km). As the flow scale is reduced, first rotation (at $\approx 10^2$ km) and then also stratification (at ≈ 1 km) become of secondary importance. Understanding the transitions between different regimes is crucial in order to evaluate the global circulating models which nowadays start to resolve them. We mainly focus on how energy is transferred from the large scales, at which it is injected, to the small-scales, where it is dissipated, in strongly rotating and stratified systems by means of numerical simulations of the Boussinesq equations. The large resolution employed, $N_x = N_y = N_z = 1024$, allows us to resolve more than one dynamical regime. Large scale dynamic closely resembles quasi-geostrophic dynamics. However, departure from a quasi-geostrophic regime may also be recognized. We show the presence of a leakage of energy which starts from the largest scales and it is entirely supported by a non-geostrophic dynamics, which is possibly stratified turbulence. Despite the idealized set considered in the study, the results surprisingly agree with observations in the atmosphere, suggesting that the presented mechanism may play a crucial role in geophysical dynamics.

[1]Linné FLOW Centre

8:52AM M7.00005 A model for turbulence in moderately stratified flows[1] , JENNIFER JEFFERSON, CHRIS REHMANN, Iowa State University — Models based on the Reynolds-averaged Navier-Stokes (RANS) equations have successfully predicted turbulence in weakly stratified flows, but they require adjustment in more strongly stratified flows to account for the interaction between turbulence and internal waves. In contrast, rapid distortion theory (RDT), which strictly applies for infinite Richardson number, captures many features of the wave mode of strongly stratified flows. To develop a model for turbulence in the moderate stratification observed in many environmental flows, we incorporated aspects of RANS models into RDT and attempted to predict the results of laboratory experiments of homogeneous turbulence in a stratified flow. Using a turbulent viscosity and diffusivity computed from length and velocity scales of the turbulence did not reproduce the timing of the oscillations of the vertical density flux. The inadequacy of the turbulent viscosity approach suggests including nonlinear interactions in the model by following the approach of Kevlahan and Hunt (1997).

[1]Funding provided by the National Science Foundation through grant 1034221.

9:05AM M7.00006 Evolution of deep-cycle turbulence in an Equatorial Undercurrent Model , HIEU PHAM, SUTANU SARKAR, KRAIG WINTERS, UC San Diego — Large-Eddy Simulation is used to investigate the relationship between the near-N oscillations and the deep-cycle turbulence in the Equatorial Undercurrent. The profiles of velocity and density in the model are similar to those observed in the field. A constant wind stress and a diurnal heat flux are applied at the surface. The model is simulated for a 2-day duration. During the day time, the wind accelerates the surface water increasing the surface shear but the turbulence intensity is low due to heating. In the evening, when the heat flux becomes neutral, shear instabilities develop in the surface layer and generate turbulence. At night time, convection due to surface cooling creates a well-mixed layer. Later at night when the convection subsides, shear instabilities grow at the base of the mixed-layer where the local gradient Richardson number falls below the critical value of 0.25. The evolution of the shear instabilities includes the temporal fluctuations of the isopycnals as well as turbulent mixing due to coherent eddies. The turbulence extends well below the surface mixed layer and lasts for a few hours. Result from our model suggests that the oscillations and the deep-cycle turbulence are related to a shear instability local to the base of the mixed layer.

9:18AM M7.00007 Parameterization of turbulent diffusivity in stratified flows using microstructure observations and DNS[1] , BENJAMIN MATER, SUBHAS VENAYAGAMOORTHY, Colorado State University — In oceanic flows, the eddy diffusivity of density, K_d, is commonly approximated using the Osborn-Cox model with a constant mixing efficiency, Γ. Many have sought to improve upon the accuracy of this approach by parameterizing the variability in Γ using the buoyancy Reynolds number ($Re_B{=}\varepsilon/\nu N^2$). In this study, we point out that $Re_B{=}Fr^2 Re_L$ (where $Fr{=}\varepsilon/\ kN$ and $Re_L = k^2/\varepsilon\nu$) and is, thus, a mixed parameter that obscures explicit dependencies on the more fundamental parameters involving turbulent kinetic energy, k. Using microstructure observations, we demonstrate this non-uniqueness of Re_B and explore the independent effects of Fr and Re_L. Because k is not readily available from microstructure measurements, however, we investigate alternative methods to infer its value from measured Thorpe scales, L_T. Through physical reasoning, we argue that L_T should scale with a length scale dependent on k and not solely on dissipation ε. We test this reasoning using DNS of decaying grid turbulence.

[1]Funded by the Office of Naval Research

9:31AM M7.00008 Turbulent Viscosity in Ekman Flow , CEDRICK ANSORGE, JUAN PEDRO MELLADO, Max Planck Institute for Meteorology — Direct numerical simulation of neutrally stratified turbulent Ekman flow is carried out at different Reynolds numbers in the range $200 < \mathbf{Re} = \delta^+ < 700$ where δ^+ is the boundary layer (BL) thickness expressed in wall units. Even if no logarithmic layer is found yet, the data suggest that in this intermediate range of $\mathbf{Re}$ certain measures of the flow approach Re-independency. The fully resolved three-dimensional fields of the turbulent flow are used to extract vertical profiles of the reynolds stresses and vertical shear. The assumption of a constant eddy viscosity over a wide range of the turbulent portion of the BL is valid with only small deviations. On the contrary, the data show that on a rotating plane (f-plane) the vectors of shear and vertical stress flux are not aligned. Hence, the assumption that the eddy viscosity is a linear function of the shear is not valid. It turns out that up to small deviations the directional offset of the shear and stress vectors is constant with height and, if varying at all, a function of $\mathbf{Re}$. This makes it possible to account for the directional offset between stress and shear in turbulence closures.

Tuesday, November 20, 2012 8:00AM - 10:10AM –

Session M8 Drops XI 25A - Chair: Arun Ramchandran, University of Toronto

8:00AM M8.00001 Spontaneous Capillarity-Driven Droplet Ejection[1] , DREW WOLLMAN, Portland State University, TREVOR SNYDER, Xerox, Wilsonville, DONALD PETTIT, NASA Johnson Space Center, MARK WEISLOGEL, Portland State University — The first large length-scale capillary rise experiments were conducted by R. Siegel fifty years ago using a drop tower at NASA LeRC. Siegel was curious if the wetting fluid would expel itself from the end of short capillary tubes in low-gravity. He observed that although the fluid partially left the tubes, it was always pulled back by surface tension, which caused it to remain pinned at the tubes' end. By exploiting tube geometry and fluid properties, we demonstrate that such capillary flows can in fact 'auto-eject' a variety of jets and drops. Multiple and stationary drops, encapsulations, and a wide range of deployed drop diameters are demonstrated using a drop tower (diameters up to ∼10mm). Terrestrial gravity experiments are demonstrated as well as droplets ejected aboard the International Space Station—drops one million times larger than their 1-g counterparts. Scaling arguments reveal the single dimensionless group that best identifies the ejection criteria. The general auto-ejection approach provides a novel mechanism from which to investigate jets, droplets, bubbles, and other large length-scale capillary phenomena.

[1]NASA NNX09AP66A: GRC, NASA NNX10AK68H: Oregon Space Grant Consortium

8:13AM M8.00002 The effect of interfacial slip on the rheology of a dilute emulsion of drops for small capillary numbers , ARUN RAMCHANDRAN, University of Toronto, L. GARY LEAL, University of California at Santa Barbara — We present the constitutive equation for the volume-averaged hydrodynamic stress for a dilute emulsion in a linear ambient flow, when there is slip at the liquid-liquid interface between the Newtonian drop and suspending fluids. Slip is modeled using the Navier slip boundary condition. We provide analytical solutions in the limit of small capillary numbers for the shape deformation, viscosity and normal stresses. Slip moderates these quantities, with changes from the no slip case being more pronounced for large drop viscosities relative to the suspending fluid. It has been suggested in the past that slip can explain the anomalously low viscosities of certain polymeric blends. Our analysis indicates that slip can only partially account for these deviations, and that other mechanisms should be explored to explain this discrepancy.

8:26AM M8.00003 Droplet generation at the critical Weber number , LAURENT TANGUY, University of Freiburg - IMTEK - Department of Microsystems Engineering, DONG LIANG, HSG-IMIT, ROLAND ZENGERLE, PETER KOLTAY, University of Freiburg - IMTEK - Department of Microsystems Engineering — The ejection of liquid droplets from a nozzle is highly important for physics of fluid. The Weber number describes how much kinetic energy is needed to overcome the surface tension and create a free-flying droplet. According to literature Weber numbers above 12 assure the creation and safe break up of a liquid droplet. However, even when this number goes down below 8, it is still possible to observe droplet break-up but with particular effects. We present here experimental results and CFD simulations for droplet break-up at low Weber number where the droplet is generated with negative kinetic energy. Such droplet generation is characterized by the droplet breaking up and then returning back into the nozzle. This is due to the fact that during the droplet formation the surface tension begins to slow down the flow velocity inside the droplet and then finally inverts the flow direction, while the droplet tail still breaks off from the nozzle. Thus after the break up the droplet momentum is oriented toward the nozzle. It is therefore possible to observe the droplet returning into the bulk fluid. High-speed images of this particular phenomenon are shown and simulation results are presented to illustrate the break up dynamics and the local velocities in the droplet.

8:39AM M8.00004 How to freeze drop oscillations with powders , JEREMY MARSTON, YING ZHU, IVAN VAKARELSKI, SIGURDUR THORODDSEN, King Abdullah University of Science and Technology — We present experiments that show when a water drop impacts onto a bed of fine, hydrophobic powder, the final form of the drop can be very different from the spherical form with which it impacts. For all drop impact speeds, the drop rebounds due to the hydrophobic nature of the powder. However, we observe that above a critical impact speed, the drop undergoes a permanent deformation to a highly non-spherical shape with a complete coverage of powder, thus creating a deformed liquid marble. This powder coating acts to freeze the drop oscillations during rebound.

8:52AM M8.00005 Flow induced by membraneless osmosis between a droplet and a bath , MATTHIEU ROCHE, CORENTIN TRÉGOUET, BO SUN, HOWARD A. STONE, Department of Mechanical and Aerospace Engineering, Princeton University — Biological material can be sorted using aqueous solutions containing two polymers. These mixtures, known as aqueous two-phase systems, experience phase separation after production because of the incompatibility of one polymer with the other. Each phase of the final solution is rich in one of the polymers. However, the concentration in polymers will often be very different from the concentration of the two initial solutions. Indeed, the interaction of water with each of the polymers induces a gradient of osmotic pressure, which leads to the redistribution of water between the two phases. Here we study the consequences of this so-called membrane-less osmosis on the deposition of a droplet of a solution of dextran on the surface of a layer of solution of poly(ethylene glycol). A flow on a scale comparable to that of the layer sets in over timescales of a few hours. We use particle tracking to describe the flow pattern. We observed the growth of vortices. By changing the concentration in polymers, the depth of the layer and the size of the drop, we relate the properties of the velocity field to the osmotic pressure of water in the ternary system water/dextran/poly(ethylene glycol). Finally, we describe important consequences of these observations for applications.

9:05AM M8.00006 Universal deformation of soft substrates near contact line reveals solid surface stresses , ROBERT STYLE, JOHN WETTLAUFER, Yale University, LARRY WILEN, Unilever, ERIC DUFRESNE, Yale University — We study how sessile droplets behave on soft substrates. Using confocal microscopy, we investigate how droplet surface tension (and Laplace pressure) deforms the substrate. We show that the near-tip shape of the wetting ridge is entirely determined by the surface tensions of the three contacting phases. In particular we can use this observation to (i) directly measure solid-vapour and solid-liquid surface tensions, (ii) resolve how out-of-plane force balance is ensured at the contact line.

9:18AM M8.00007 The stability of viscous liquid filaments , THEO DRIESSEN, University of Twente, ROGER JEURISSEN, Eindhoven University of Technology, HERMAN WIJSHOFF, OCE technologies N.V. , DETLEF LOHSE, University of Twente — The stability of liquid filaments is relevant both in industrial applications, such as inkjet printing and atomization, and in nature, where the stability of filaments has a large influence on the final drop size distribution of rain droplets and waterfalls. The liquid filament may either stably collapse into a single droplet, or break up into multiple droplets. Which scenario is realized depends on the viscosity and the aspect ratio of the filament. Here we study the collapse of an axisymmetric liquid filament is analytically and with a numerical model. We find that a long, high viscous filament can only break up due to the Rayleigh-Plateau instability, whereas a low viscous filament can break up due to end-pinching. The theory shows quantitative agreement with recent experimental findings by Castréjon-Pita et al., PRL 108, 074506 (2012).

9:31AM M8.00008 Contraction dynamics of planar liquid filaments , NICOLE DEVLIN, KRISHNARAJ SAMBATH, MICHAEL HARRIS, OSMAN BASARAN, School of Chemical Engineering, Purdue University — Thin liquid sheets are ubiquitous in nature and urban landscapes, e.g. waterfalls, and industry, e.g. in various atomizers where sheets of liquid emanate from a nozzle or off a solid surface. These liquid sheets contract due to surface tension and may or may not break into smaller fragments depending on physical properties and flow conditions. The cross-section of a liquid sheet in a plane perpendicular to the main flow direction is a planar or 2D filament. Here, we study the contraction dynamics of an idealized 2D filament of an incompressible Newtonian fluid the initial shape of which is a rectangle terminated by two identical semi-circles. The dynamics are analyzed by solving the full 2D Navier-Stokes system and a1D, slender-jet approximation to it by a numerical technique based on the Galerkin finite element method. Simulation results are summarized by means of a phase diagram in the space of Reynolds number and initial filament aspect ratio. The talk will conclude with a discussion of the different modes of contraction and a critique of the capabilities and limitations of the 1D model.

9:44AM M8.00009 Disk impact on an oil-water bilayer , DEVARAJ VAN DER MEER, IVO PETERS, MATTEO MADONIA, DETLEF LOHSE, University of Twente — When a round disc impacts on a water surface, a cavity is created that collapses under the influence of hydrostatic pressure. This leads to the formation of two jets that originate from the pinch-off point, one shooting upwards, the other downwards. To investigate the mechanism by which the jets are formed, we add a thin layer of oil on top of the water surface and repeat the experiment. Initially, just after the pinch-off of the cavity, the jet consists exclusively of oil. At a later stage, we observe that water enters into the jet, but a stable core of oil remains that extends all the way down into the bulk of the liquid. We compare our findings to several theoretical models from the literature.

9:57AM M8.00010 Mesler entrainment in alcohols , J.R. SAYLOR, R.K. SUNDBERG, Clemson University — When a drop impacts a flat surface of the same liquid at an intermediate velocity, the impact can result in the formation of a very large number of very small bubbles. At lower velocities, drops bounce or float, and at larger velocities a single bubble forms, or there is a splash. The formation of large numbers of small bubbles during intermediate velocity impacts is termed Mesler entrainment and its controlling mechanism is poorly understood. Existing research has shown that Mesler entrainment is highly irreproducible when water is the working fluid, and very reproducible when silicone oil is the working fluid. Whether this is because water is problematic, or silicone oil is uniquely well-suited, is unclear. To answer this question, experiments were conducted using three different alcohols. The results of these experiments were very reproducible for all alcohols tested, suggesting that there is something unique about water which accounts for its lack of reproducibility. The data from these experiments were also used to develop a dimensionless group that quantifies the conditions under which Mesler entrainment occurs. This dimensionless group is used to provide insight into the mechanism of this unique method of bubble formation.

Tuesday, November 20, 2012 8:00AM - 10:10AM –

Session M9 Interfacial/Thin Film Instability VI 25B - Chair: Mohamed Gad-el-Hak, Virginia Commonwealth University

8:00AM M9.00001 Electrohydrodynamic instabilities in thin viscoelastic films: AC and DC fields , LEONARDO ESPIN, ANDREW CORBETT, SATISH KUMAR, Department of Chemical Engineering and Materials Science, University of Minnesota, KUMAR RESEARCH GROUP TEAM — Electrohydrodynamic instabilities in thin liquid films are a promising route for the self-assembly of well-defined topographical features on the surfaces of materials. Here, we study the effect of viscoelasticity on these instabilities under the influence of AC and DC electric fields. Viscoelasticity is incorporated via a Jeffreys model and both perfect and leaky dielectric materials are considered. In the case of DC fields, asymptotic methods are employed to shed light on the nature of a singularity that arises when solvent viscosity is neglected (i.e., the Maxwell-fluid limit). In the case of AC fields, we apply a numerical procedure based on Floquet theory to determine the maximum growth rate and corresponding wavenumber as a function of the oscillation amplitude and frequency. Elasticity is found to increase both the maximum growth rate and the corresponding wavenumber, with the effects being the most pronounced when the oscillation period is comparable to the fluid relaxation time.

8:13AM M9.00002 Dynamics of a thin ferrofluid film subjected to a magnetic field[1] , DEVIN CONROY, ALEX WRAY, DEMETRIOS PAPAGEORGIOU, RICHARD CRASTER, 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; in this work, however, we take the limit of small Langevin parameters, in which this function becomes linear. We use the long-wave limit to expand the governing equations associated with the film; no such approximation is applied in the gas phase in which the full Laplacian for the potential is solved. A one-dimensional partial differential equation is then derived that governs the nonlinear evolution of the interface. This equation is solved numerically for a wide range of system parameters. The results of this parametric study will be presented.

[1]EPSRC DTA

8:26AM M9.00003 Nonlinear traveling waves in confined ferrofluids[1] , SÉRGIO LIRA, JOSÉ MIRANDA, Universidade Federal de Pernambuco — We study the development of nonlinear traveling waves 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 employ a mode-coupling theory which predicts the possibility of controlling the speed of the waves by purely magnetic means. The influence of the tilted magnetic field on the waves shape profile, and the establishment of stationary traveling wave structures are investigated.

[1]We thank CNPq (Brazilian Research Council) for financial support.

8:39AM M9.00004 Sustainability of Superhydrophobicity Under Pressure , HOOMAN VAHEDI TAFRESHI, MOHAMED A. SAMAHA, MOHAMED GAD-EL-HAK, Department of Mechanical & Nuclear Engineering, Virginia Commonwealth University, Richmond, VA 23284 — Prior studies have demonstrated that superhydrophobicity of submerged surfaces is influenced by hydrostatic pressure and other environmental effects. Sustainability of a superhydrophobic surface could be characterized by both how long it maintains the trapped air in its surface pores, so-called "longevity," and the pressure beyond which it undergoes a global wetting transition, so-called "terminal pressure." In this work, we investigate the effects of pressure on the performance of electrospun polystyrene fibrous coatings. The time-dependent hydrophobicity of the submerged coating in a pressure vessel is optically measured under elevated pressures, up to 10 bar. Rheological studies are also performed to determine the effects of pressure on drag reduction and slip length. The measurements indicate that surface longevity exponentially decays with increasing pressure in perfect agreement with prior studies conducted at much lower pressures. It is found, however, that fibrous coatings could resist hydrostatic pressures significantly higher than those of previously reported surfaces. Our observations indicate that superhydrophobic fibrous coatings could potentially be used for underwater applications.

8:52AM M9.00005 Effects of Hydrostatic Pressure on the Drag Reduction of Submerged Aerogel-Particle Coatings , MOHAMED GAD-EL-HAK, HOOMAN VAHEDI TAFRESHI, MOHAMED A. SAMAHA, Department of Mechanical & Nuclear Engineering, Virginia Commonwealth University, Richmond, VA 23284 — Hydrophobic aerogel particles with different average diameters are randomly deposited onto metallic substrates with a thin adhesive coating to achieve a combination of hydrophobicity and surface roughness. The resulting surfaces show different degrees of superhydrophobicity and are used to study the effects of elevated pressure on the drag reduction and the degree of hydrophobicity (sustainability) of such surfaces when used for underwater applications. We also developed an image-thresholding technique to estimate the gas area fraction of the coating. The results indicate that there exists a new parameter, the terminal pressure, beyond which the surface undergoes a global transition from the Cassie state to the Wenzel state, and therefore can no longer generate drag reduction. This terminal pressure differs from the previously identified critical pressure. The latter is the pressure above which the surface starts the transition process at some location, but not necessarily at other spots due to the heterogeneity of the surface. For the particle coatings used herein, the terminal pressures are measured to range from 100 to 600 kPa, indicating that such coatings could potentially be used for deep underwater applications.

9:05AM M9.00006 Dynamics of an inclined film in the presence of soluble surfactants[1] , ANNA GEORGANTAKI, GEORGE KARAPETSAS, VASILIS BONTOZOGLOU, University of Volos — We investigate the dynamics of a thin film flowing down an inclined solid surface in the presence of soluble surfactants. Lubrication theory for the fluid motion, and advection-diffusion equations as well as chemical kinetic fluxes for the surfactant transport, lead to coupled evolution equations for the film thickness, interfacial concentrations of surfactant monomers and bulk concentrations of monomers. We solve numerically the evolution equations using the finite element method and we perform a full parametric study. The results of our simulations show that surfactants have a strong stabilizing effect on the flow due to the presence of Marangoni stresses. The wave patterns that arise differ significantly from the case of clean fluids. It will be shown that the dominant structures, even at high Re numbers, are sinusoidal traveling waves in direct agreement with experimental observations.

[1]The financial support by the GSRT Greece (Grant No: PE8(908)) is thankfully acknowledged.

9:18AM M9.00007 Thin soap films are quasi-2D fluids and thick soap films are not , SKANDA VIVEK, ERIC R. WEEKS, Physics Department, Emory University — We use microrheology to measure the 2D (interfacial) viscosity of soap films. Microrheology uses the diffusive motion of tracer particles suspended in the soap film to infer the viscosity. Our particles are colloids of diameter $d = 0.5$ μm. We measure the interfacial viscosity of soap films ranging in thickness from $h = 0.5$ μm to 2.0 μm. The thickness of these films is measured using the infrared absorbance of the water based soap films, based on a previous setup [X. L. Wu, R. Levine, M. A. Rutgers, H. Kellay, W.I. Goldburg, *Rev. Sci. Inst.* **72**, 2467 (2001)]. From the knowledge of the film thickness and the viscosity of the fluid used to make the film, we can infer the interfacial viscosity due to the surfactant layers at the film/air interfaces. Consistent results are found for thin films ($h/d < 3$) whereas for thicker films inconsistent and unphysical results are found indicating 3D effects begin to play a role. The transition from 2D to 3D properties as a function of h/d is sharp.

9:31AM M9.00008 Thin film dynamics of viscoelastic fluids , LUC LEBON, LAURENT LIMAT, CNRS / Univ. Paris Diderot — We present here viscoelastic fluids in thin film flows, such as liquid bells or liquid curtains. The viscoelastic property of the liquids exhibits specific dynamics in such flows. In the case of bells, the elastic strength tends to extend the bell size for example. In the case of curtain flows, original behaviour of holes are observed with specific growth mechanism for bubbles trapped in the flow.

9:44AM M9.00009 Fluid displacement under elastic membranes: Dynamics and interfacial instabilities[1] , TALAL AL-HOUSSEINY, IVAN CHRISTOV, Princeton University, ANNE JUEL, University of Manchester, HOWARD STONE, Princeton University — The spreading of fluids under a flexible membrane is a feature of many systems such as the lateral intrusion of magma under a terrestrial crust, or when blood spreads underneath the skin giving the signature color of bruises. In this work, we investigate the displacement of a viscous fluid by a gas underneath an elastic membrane. We consider a radial Hele-Shaw cell where the upper plate is an elastic sheet. The dynamics of the interface between the injected gas and the displaced fluid are fundamentally modified by the presence of an elastic boundary, which leads to the suppression of viscous fingering below a critical flow rate. We demonstrate theoretically the mechanism of suppression and find the corresponding critical flow rate. In addition, we study the dynamics of a stable (circular) interface propagating underneath an elastic membrane and derive the scaling laws for both the position of the interface and the shape of the elastic membrane. Our theoretical findings agree very well with the experimental results of D. Pihler-Puzovic et al. (PRL 2012).

[1]T. T. Al-Housseiny is supported by the National Science Foundation Graduate Research Fellowship under Grant No. DGE-0646086.

9:57AM M9.00010 Elastically-driven surface plumes in rimming flow of a non-Newtonian fluid , GABRIEL SEIDEN, VICTOR STEINBERG, Department of Physics of Complex Systems, Weizmann Institute of Science, Rehovot 76100, Israel — A polymer solution partially filling a rotating horizontal drum undergoes an elastically driven instability at low Reynolds numbers. This instability manifests itself through localized plumelike bursts, perturbing the free liquid surface. We present experimental results on the dynamics of individual plumes and the statistics pertaining to the complex collective interaction between plumes, which leads to plume coagulation.

Tuesday, November 20, 2012 8:00AM - 10:10AM –

Session M10 General Instability I 25C - Chair: Brent Houchens, Rice University

8:00AM M10.00001 Stability of Miscible Displacements in Porous Media for Time-Dependent Injection Velocities[1] , QINGWANG YUAN, JALEL AZAIEZ, Department of Chemical and Petroleum Engineering, University of Calgary — Flow instabilities are often observed when a high-viscosity fluid is displaced by a low-viscosity fluid in porous media. Even though most existing studies have analyzed such instabilities in the case of constant injection velocity, in many practical processes such as ground water flows and enhanced oil recovery, the velocity is actually time dependent. This work presents linear stability analysis of miscible displacement with time-dependent injection velocities in rectilinear Hele-Shaw cell. Both quasi-steady-state approximation (QSSA) and initial value calculation (IVC) methods are used to analyze the stability of constant, sinusoidal, and step injection velocity models. For QSSA, it is found that the growth rate follows the behavior of the injection velocity, while it also tends to flatten out whenever the contribution to the velocity is negative. For such negative velocities, the inverse displacement is unconditionally stable. For IVC, the variation of growth rate is influenced by velocity amplitude Γ and time. The growth rates at maximum and minimum velocities are nearly the same as their constant velocity counterparts when Γ is small. For large Γ, they are different, with some special characteristics.

[1]China Scholarship Council (CSC); Natural Sciences and Engineering Research Counsel (NSERC); Western Canada Research Grid (WestGrid).

8:13AM M10.00002 Stability and sensitivity analyses of the engulfment regime in a three dimensional T-shaped micromixer , ANDREA FANI, SIMONE CAMARRI, Dep. of Aerospace Engineering, University of Pisa, Italy, CHIARA GALLETTI, Dep. of Chemical Engineering, University of Pisa, Italy, MARIA VITTORIA SALVETTI, Dep. of Aerospace Engineering, University of Pisa, Italy — The recent research in micro-fluidics has focused on the development of efficient passive micromixers, in which mixing is promoted without the help of any external power. One among the simplest designs of a passive micromixer is a T shape, in which the inlets join the main channel with T-shaped branches. The range of Reynolds numbers, Re, of interest for practical applications is such that the flow inside such a mixer is laminar but it is characterized by peculiar fluid-dynamics instabilities, which significantly enhance mixing but are poorly investigated in the literature. As Re is increased, the flow goes through a bifurcation which drives the system from a perfectly symmetric flow to a steady but asymmetric state, so enhancing mixing (engulfment regime). The onset of the engulfment has been found to be influenced by geometrical parameters and by inflow conditions. In the present work we characterize the engulfment instability by a global stability analysis on the 3D base flow in a T-mixer. Sensitivity analyses with respect to a structural perturbation of the linearized flow equations and to a base flow modification were carried out. Finally, we characterize the sensitivity of the considered instability with respect to a perturbation of the inlet velocity profile.

8:26AM M10.00003 Investigation of Linear Stability Theory for Wavy Interface in Magnetic Pulse Welding , ALI NASSIRI, GREGORY CHINI, BRAD KINSEY, University of New Hampshire — 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. MPW is a lap welding method: the two workpieces are placed in a roughly parallel configuration with a small gap between them to achieve high impact velocity and pressure. Intriguingly, 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. The mechanism underlying this wavy pattern is still not well understand. 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 an energy source for the vortical pattern. 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 a perfectly plastic solid material. Preliminary results confirm that an instability giving rise to a wavy interfacial pattern is possible.

8:39AM M10.00004 Analysis of Float-Zone Crystal Growth Instabilities Through Linear Stability Analysis and 3D Spectral Element Simulations , BRENT HOUCHENS, KENNETH DAVIS, Rice University, YUE HUANG, ExxonMobil — In the optically-heated floating zone crystal growth process, a region of polycrystalline rod is melted and resolidified as a single crystal. Increased heat flux causes the axisymmetric base flow to transition to a fully three-dimensional flow, leading to various defects in the grown crystals. Here we use linear stability analysis to determine the point of initial instability and to compute the eigenfunctions corresponding to the unstable growth modes. In addition, three-dimensional, time-dependent spectral element simulations are computed and the results for the initial bifurcation point as well as the flow field are then compared to the results from linear stability theory. Simulations beyond the initial instability show competition between two stationary modes, which mimic a quasi-periodic mode. Additionally, the melt region is subjected to an axial magnetic field to resist velocities in the radial and azimuthal directions, stabilizing the base state. Simulations and linear stability analyses are compared for cases with the applied magnetic field and both confirm the desired stabilization.

8:52AM M10.00005 Sheared Electroconvective Instability , RHOKYUN KWAK, Massachusetts Institute of Technology, VAN SANG PHAM, KIANG MENG LIM, Singapore-MIT Alliance, National University of Singapore, Singapore, JONGYOON HAN, Massachusetts Institute of Technology — Recently, ion concentration polarization (ICP) and related phenomena draw attention from physicists, due to its importance in understanding electrochemical systems. Researchers have been actively studying, but the complexity of this multiscale, multiphysics phenomenon has been limitation for gaining a detailed picture. Here, we consider electroconvective(EC) instability initiated by ICP under pressure-driven flow, a scenario often found in electrochemical desalinations. Combining scaling analysis, experiment, and numerical modeling, we reveal unique behaviors of sheared EC: unidirectional vortex structures, its size selection and vortex propagation. Selected by balancing the external pressure gradient and the electric body force, which generates Hagen–Poiseuille(HP) flow and vortical EC, the dimensionless EC thickness scales as $(\phi^2/U_{\mathrm{HP}})^{1/3}$. The pressure-driven flow(or shear) suppresses unfavorably-directed vortices, and simultaneously pushes favorably-directed vortices with constant speed, which is linearly proportional to the total shear of HP flow. This is the first systematic characterization of sheared EC, which has significant implications on the optimization of electrodialysis and other electrochemical systems.

9:05AM M10.00006 A localized relaxation scheme for the computation of steady flows , JEAN-MARC CHOMAZ, XAVIER GARNAUD, LadHyX, Ecole Polytechnique — The computation of steady flow solutions in unstable settings is often the first step in studying the instability features. For this purpose, we present a method inspired by the Selective Frequency Damping of Akervik *et al.* (2006). A low-pass temporal filter is applied at a small number of locations in the flow, and these values are used to relax the nonlinear system. If the relaxation points are properly selected, such a scheme may stabilize the dynamics. In this case, the steady flow can be computed using a regular time marching procedure with almost the same computational cost and memory requirements as a regular simulation. The relation between the optimal location of the relaxation points and the wavemaker region will be discussed.

9:18AM M10.00007 A spectral representation of oscillators , SHERVIN BAGHERI, Linné Flow Centre, KTH Mechanics, Stockholm — We investigate Koopman modes as an expansion basis for describing the nonlinear dynamics of self-sustained oscillating flows. The method decomposes the flow evolving on a limit cycle and on its stable manifold into asymptotic/transient and steady/oscillatory components, providing an accurate prediction of both frequencies and relaxation rates. We find that the leading Koopman modes of an oscillator correspond to the mean flow, shift modes and nonlinear global modes. Close to the critical bifurcation threshold the modes are explicitly formed using multiple scale expansion of the flow field and a spectral expansion of the corresponding amplitudes. The analytic modes are in good agreement with Ritz vectors obtained computationally using the dynamic mode decomposition algorithm. We further discuss the ability of the Koopman modes and Ritz vectors of a nonlinear system to approximate the dynamics of unstable equilibria and the transient dynamics characterized by non-exponential behavior.

9:31AM M10.00008 Criterion of Turbulent Transition in Pressure Driven Flows , HUA-SHU DOU, Zhejiang Sci-Tech University, BOO CHEONG KHOO, National University of Singapore — It has been found from numerical simulations and experiments that velocity inflection could result in turbulent transition in viscous parallel flows. However, there are exceptions, for example, in the plane Poiseuille-Couette flow. Thus, whether velocity inflection necessarily leads to turbulent transition is still not clear. To-date, there is still no consensus on the physics of turbulence transition in the scientific community. In this study, the mechanism of turbulent transition is investigated using the energy gradient method. It is found that the transition to turbulence from a laminar flow depends on the magnitudes of the energy gradient function and the energy of the disturbance imposed (including both the amplitude and the frequency). Our study further reveals that the criterion of turbulent transition is different in pressure and shear driven flows. In pressure driven parallel flows, it is found that the necessary and sufficient condition of turbulent transition is the existence of an inflection point on the velocity profile. This criterion is found to be consistent with the available experimental data and numerical simulation results. On contrast, velocity inflection in shear driven flows does not necessarily lead to turbulent transition.

9:44AM M10.00009 Experimental Study of Transition to Turbulence in a Kolmogorov-Like Flow[1] , BALACHANDRA SURI, JEFFREY TITHOF, RADFORD MITCHELL, 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. Here, we present results from an experimental implementation of a Kolmogorov-like flow where a thin layer of electrolyte is driven electromagnetically. Using PIV to extract the velocity fields, we quantitatively study the bifurcations that the system undergoes as it transitions to turbulence. These results are in good quantitative agreement with those from a direct numerical simulation of a two-dimensional flow model. We also discuss our on-going work on identifying ECS in these flows and studying their role in the weakly turbulent regime.

[1]This work is supported under NSF grant CBET-0853691.

9:57AM M10.00010 On the added-mass effects of mean compressible and incompressible flows in fluid-solid interaction , R. JAIMAN, National University of Singapore, M. PARMAR, University of Florida, PARDHA SARADHI, National University of Singapore — The unsteady fluid-structure interaction (FSI) is of fundamental interest in its own right. It is also of practical interest in a wide range of natural phenomena and industrial applications. Understanding the effects of flexible wall and estimates of added-mass forces can provide insights to coupled fluid-structure dynamics as well the improvements of numerical algorithms and closed-form empirical relations. Starting from the seminal work of Kramer (M.O. Kramer. Readers Forum, J. Aerospace Sci., 27(68), 1960.) numerous work have been published describing experiments, theoretical analysis and computations to understand how flexible elastic walls affect hydrodynamic stability. Most of the prior analysis focused on inviscid incompressible flow interacting with an elastic flat plate. In this work we present further theoretical investigation of the added-mass effect and the instability of an elastic plate as well as a string under a variety of uniform mean flows consisting of incompressible and compressible, inviscid and viscous conditions. We discuss the influence of viscosity and compressibility on the instability modes of fluid-solid interaction. We note that the added-mass effect for incompressible flow has a global character and for compressible flow it is time dependent.

Tuesday, November 20, 2012 8:00AM - 10:10AM –

Session M11 Viscous Flows 26A - Chair: Ken Kamrin, Massachusetts Institute of Technology

8:00AM M11.00001 Viscous Added Mass of a Moving Solid Object in a Closed Liquid-Filled Container , J.R. TORCZYNSKI, L.A. ROMERO, Sandia National Laboratories — A moving solid object in a closed liquid-filled container is shown to have a viscous added mass in the quasi-steady Stokes limit. The viscous added mass is similar to the added mass for potential flow. The added-mass force is the product of the viscous added mass and the object's acceleration and is analogous to but distinct from the drag force, which is the product of the drag coefficient and the object's velocity. Both the drag coefficient and the viscous added mass can be computed directly from the quasi-steady Stokes solution for the liquid velocity field. The viscous added mass arises from the fact that the object's acceleration changes the kinetic energy of the liquid as well as the object. If the object fills most of the container's cross section, the viscous added mass is much larger than the object's mass and thus is dynamically significant. Sandia National Laboratories is a multi-program laboratory managed and operated by Sandia Corporation, a wholly owned subsidiary of Lockheed Martin Corporation, for the U.S. Department of Energy's National Nuclear Security Administration under contract DE-AC04-94AL85000.

8:13AM M11.00002 The effect of the torsion on the axial flow in helical pipes , DOMENIC D'AMBROSIO, MASSIMO GERMANO, Politecnico di Torino - Italy — The effect of the torsion on the flow in helical pipes remains difficult to predict and slightly controversial. Also in the simple case of a circular cross section and laminar flow it is difficult to have a clear idea of the influence of the torsion. In particular as regards the axial flow we register a second order effect of the torsion on the flow rate but a strong effect on the asymmetry of the velocity profile produced by the curvature. In the presentation a new bulk indicator of the rotation of the axial flow induced by the torsion is defined. This indicator is applied to the accurate calculations of Gammack and Hydon, (JFM 433, 357-382, (2001)), and an estimate of the bulk rotation angle of the axial flow is produced in the limiting case of low curvature and high values of the torsion. In this case we register a strong and direct dependence of the bulk rotation of the axial flow with the torsion number.

8:26AM M11.00003 ABSTRACT WITHDRAWN –

8:39AM M11.00004 Flow in and geometry of microstructured optical fibres , YVONNE STOKES, The University of Adelaide, DARREN CROWDY, Imperial College London, HAYDEN TRONNOLONE, HEIKE EBENDORFF-HEIDEPRIEM, The University of Adelaide — Microstructured optical fibres (MOFs) have revolutionised optical fibre technology, promising a virtually limitless range of fibre designs for a wide range of applications. Extrusion of a preform and drawing to form a fibre is a promising fabrication process for mass production. However, understanding of the flow during fabrication and its effect on the complex air-solid structure in the MOF cross section is lacking, and this impedes MOF development. We propose a modelling methodology suitable for complex structure, and focus on flow in the cross section during preform extrusion. Excellent qualitative agreement of model results and experiment is shown and areas for model improvement are identified.

8:52AM M11.00005 The moving contact line problem. Is there a solution? , DAVID SIBLEY, ANDREAS NOLD, NIKOS SAVVA, SERAFIM KALLIADASIS, Department of Chemical Engineering, Imperial College London — The moving contact line problem occurs when one fluid replaces another as it moves along a solid surface, a situation arising in a vast range of applications. The apparent paradox of motion of the fluid-fluid interface, yet static fluid velocity at the solid satisfying the no-slip boundary condition, has been known for a number of decades, with a wealth of publications suggesting methods to resolve it since. Here we consider the behaviour of a solid-liquid-gas system near the three-phase contact line using a diffuse-interface model with no-slip at the solid and where the fluid phase is specified by a continuous density field. We first obtain a wetting boundary condition on the solid that allows us to consider the motion without any additional physics. Careful examination then of the asymptotic behaviour as the contact line is approached is shown to resolve the moving contact line problem. Various features of the model are scrutinised alongside extensions to incorporate slip, finite-time relaxation of the chemical potential, or a precursor film at the wall. But these are not necessary to resolve the moving contact line problem.

9:05AM M11.00006 Using Simple Flows to Tie Knots in Flexible Fibers , STEVE KUEI, Princeton University, CHRIS SADLEJ[1], Institute of Fundamental Technological Research, Polish Academy of Sciences, HOWARD A. STONE, Princeton University — Flexible fibers, such as DNA and other polymer chains, have sometimes been found to contain knotted regions. While such fibers are not strict, closed knots, they exhibit similar characteristics; the formation of these "open knots" and the effects they have on material properties are the subject of current research. We investigate the possibility that simple flows can generate open knots in sufficiently long and flexible elastic fibers. Using the HYDROMULTIPOLE algorithm, which solves the multipole expansion of Stokes equations, we use numerical simulations to study the time evolution of a bead-spring model fiber in a shear flow. The model is described by the dimensionless parameters k, A, and l_0, which account for the elastic forces, bending forces, and equilibrium distance between beads, respectively. We find that for certain systems, the characteristic tumbling motion of a fiber in shear flow will result in the formation of 3_1 and 5_1 knots, as identified by their Alexander polynomial knot invariants. Investigation of the knotting mechanism is ongoing.

[1]We dedicate this work to our collaborator Chris Sadlej who is recently deceased.

9:18AM M11.00007 Inlet Jet Interaction in Horizontal Pipe Flow[1] , PRANAB JHA, Dept. of Mechanical Engineering, University of Houston, CHUCK SMITH, Apache Corporation, RALPH METCALFE, Dept. of Mechanical Engineering, University of Houston — Laminar incompressible flow (Re $<$ 1000) inside a horizontal channel with multiple cross-flow inlets was studied numerically. First, two cross-flow inlets were used to observe the flow interference phenomenon between the inlets. This concept was extended to axisymmetric pipe flow 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 different inlet sizes on the flow regimes under steady state condition were studied. A hydrostatic model of fluid reservoirs draining into the channel was constructed using a linear function for pressure at the inlet boundaries to study the dynamic behavior of the inlets. Three different time scales related to the depletion of the reservoirs were identified. The dynamic behavior of two cross-flow inlets was observed with the initial conditions corresponding to the three flow regimes. Similar study was carried out for a five-inlet case and the dynamic behavior of individual reservoirs was observed. The change of flow regimes in the system over time with reservoir draining was evident and the different time-scales involved were identified.

[1]Supported in Part by Apache Corporation

9:31AM M11.00008 Streamline Patterns and Eddies in Slipping Stokes Flow , D. PALANIAPPAN, Texas A&M University - Corpus Christi — Streamline topologies are analyzed in the vicinity of boundaries in the limit of Stokes flow with Navier slip boundary conditions for some simple flows involving two- and three-dimensional configurations. It is found that the streamline pattern transformations, and consequently the flow fields are sensitive to the non-dimensional slip parameter λ. For two-dimensional flows, the separated/attached eddies - that are known to exist in the no-slip case at the contour - get destroyed or pushed away from the boundary as the slip is varied. Analysis of flow generated by a point force (stokeslet) inside a spherical container reveals that when the stokeslet is positioned at the center of the container, the eddy pattern - that is noted in the no-slip case - undergoes a series of transformations due to slip variations and eventually disappears. Furthermore, the parameter λ dictates the locations of the stagnation point and the point of zero vorticiy in the flow domain. Our analytical solution indicates that the *co-existence of a stagnation point (r_{stag}) and a point of zero vorticity ($r_{\Omega=0}$) in the flow region is necessary for the occurrence of closed eddies.* The results may be of some interest in small scale hydrodynamics in which Stokes flow occurs.

9:44AM M11.00009 Analysis of Slip Boundary Condition in Single and Multi-Phase Flows[1] , JOSEPH THALAKKOTTOR, KAMRAN MOHSENI, University of Florida — Over the past two decades several studies have been conducted to understand the molecular mechanism of slip in fluids at the boundary and to better understand the contact point singularity in two phase flow. Although for single phase flows, researchers have looked into the effects of unsteady flow in gases; in liquids, most of the study has been limited to steady flows. In this paper we use molecular dynamic simulations to study slip in an unsteady flow. An unsteady slip model is established, the non-dimensionalizing of which leads to a universal curve for boundary slip. The universal curve gives the slip length for a given shear rate and gradient of shear rate, for steady and unsteady flow. We also identify a non-dimensional number which is defined as the ratio of phase speed to local speed of sound that explains the mechanism responsible for the transition of slip boundary condition from finite to perfect slip. The slip boundary condition is further studied for steady and unsteady multi-phase flows. Emphasis is placed on observing the slip at the wall at the fluid-fluid interface. We establish a universal curve for slip boundary condition for multi-phase flow, for steady and unsteady flows.

[1]Office of Naval Research

9:57AM M11.00010 Effective slip identities for viscous flow over arbitrary patterned surfaces , KEN KAMRIN, MIT, PIERRE SIX, Ecole Nationale Superieure des Mines de Paris — For a variety of applications, most recently microfluidics, the ability to control fluid motions using surface texturing has been an area of ongoing interest. In this talk, we will develop several identities relating to the construction of effective slip boundary conditions for patterned surfaces. The effective slip measures the apparent slip of a fluid layer flowing over a patterned surface when viewing the flow far from the surface. In specific, shear flows of tall fluid layers over periodic surfaces (surfaces perturbed from a planar no-slip boundary by height and/or hydrophobicity fluctuations) are governed by an effective slip matrix that relates the vector of far-field shear stress (applied to the top of the fluid layer) to the effective slip velocity vector that emerges from the flow. Of particular note, we will demonstrate several general rules that describe the effective slip matrix: (1) that the effective slip matrix is always symmetric, (2) that the effective slip over any hydrophobically striped surface implies a family of related results for slip over other striped surfaces, and (3) that when height or hydrophobicity fluctuations are small, the slip matrix can be approximated directly using a simple formula derived from the surface pattern.

Tuesday, November 20, 2012 8:00AM - 9:57AM –

Session M12 Vortex VII 26B - Chair: Dustin Kleckner, University of Chicago

8:00AM M12.00001 Relative equilibria in rotating shallow water layer: a real fluid case of point vortex theory , MOHAMED FAYED, Department of Mechanical and Industrial Engineering, Concordia University, Montreal, Quebec, Canada, HAMID AIT ABDERRAHMANE, King Abdullah University of Science and Technology (KAUST), HOI DICK NG, GEORGIOS H. VATISTAS, Department of Mechanical and Industrial Engineering, Concordia University, Montreal, Quebec, Canada — The present work deals with the question whether or not the regular equilibrium structures, consisting of two and three vortices in rotating shallow water layer, produced inside a cylindrical container with a revolving disk at the bottom, represent real fluid cases of the old point vortex theory. Despite an attempts made by some researchers to address this question, the answer is yet to be clarified. Based on the data from our experiments we show that the observed vortex-pattern do retain the fundamental characteristics of Kevin's equilibria that can be adequately described by the classical idealized point vortex theory. Equivalently, we demonstrate that the experimental results found in recent literature, if properly interpreted, lead to the same conclusion.

8:13AM M12.00002 ABSTRACT WITHDRAWN –

8:26AM M12.00003 Regenerative centrifugal instability on a vortex column , ERIC STOUT, FAZLE HUSSAIN, University of Houston — The limitation and renewal of centrifugal instability of a vortex column (due to a sheath of negative axial vorticity, -Ω_z, surrounding the +Ω core, i.e. a circulation overshoot) is studied via the transport dynamics of perturbations to the initially unstable vortex using DNS of the incompressible Navier-Stokes equations for a range of vortex Reynolds numbers (Re=circulation/viscosity). Any radial perturbation vorticity, ω'_r, is tilted by the column's mean shear to form filaments with azimuthal vorticity, ω'_θ, generating positive Reynolds stress, $+u'v'$ (u',v' are the radial and azimuthal perturbation velocities), required for energy growth. This ω'_θ in turn tilts -Ω_z to amplify ω'_r (and consequently ω'_θ) – thus causing instability. Limitation of ω'_r growth, thus also energy production, occurs as the perturbation transports angular momentum (rV) radially outward from the overshoot, moving the overshoot outward, hence lessening and shifting -Ω_z, while also transporting core +Ω_z, around the location of the filament. After the overshoot shifts, tilting of -Ω_z reverses ω'_r (hence reducing ω'_θ), causing the filament to generate $-u'v'$, i.e. energy decay, and hence self-limitation of growth. Associated with $--u'v'$ is the filament's radially inward transport of rV, which can produce a new circulation overshoot and renewed instability. New overshoot formation and renewed generation of $+u'v'$ is examined using a helical ($m = 1$) mode – a promising scenario for regenerative transient growth and possible turbulence generation on a vortex column.

8:39AM M12.00004 Stirring vortices with vorticity holes , OSCAR VELASCO FUENTES, Departamento de Oceanografia Fisica, CICESE — A vorticity hole is a region with, in absolute value, significantly lower vorticity than its surroundings. Here we discuss the dynamics of a Rankine vortex with two equal circular holes. If a symmetric initial condition is assumed, the evolution depends on three parameters: the vorticity drop, the hole size and the distance between the holes. We computed the evolution with a contour-dynamics model and quantified the stirring of fluid particles using finite-time Lyapunov exponents and Melnikov's method. The vorticity holes evolve similarly to a pair of vortices in an otherwise quiescent fluid, although they are additionally affected by their interaction with the boundary of the Rankine vortex. The strongest stirring occurs when the holes interact elastically and then always in the center of the vortex. This result contradicts the generally accepted notion that vortices are regions of null to weak stirring.

8:52AM M12.00005 Three-dimensional vortex analysis and aeroacoustic source characterization of jet core breakdown , DANIELE VIOLATO, FULVIO SCARANO, TUDelft — The 3D patterns of jet core breakdown are investigated in a jet at *Re*=5,000 by time-resolved tomographic particle image velocimetry in the range between 0 and 10 jet diameters. The characteristic pulsatile motion of vortex ring shedding and pairing culminates with the growth of primary in-plane and out-of-plane azimuthal waves and leads to the formation of streamwise vortices. Vortex ring humps are tilted and ejected along the axial direction as they are subjected to higher axial velocities. By the end of the potential core, this process causes the breakdown of the vortex ring regime and the onset of streamwise filaments oriented at 30-45 degrees to the jet axis. A three dimensional modal analysis of velocity and vorticity fields is conducted by proper orthogonal decomposition within the first 10 modes. The decomposed velocity fluctuations describe helical motion in the region of the jet core-breakdown and, further downstream, jet axis flapping and precession motions. By the end of the potential core, vorticity modes show travelling waves of radial and axial vorticity with a characteristic 40 degree inclination to the jet axis. Following Powell's aeroacoustic analogy, the instantaneous spatial distribution of the acoustic source term is mapped. Far-field acoustic predictions are given based on the direct evaluation of Powell's analogy with the tomographic data.

9:05AM M12.00006 Partial reconnection of orthogonal vortices[1] , LOUIS DUFRESNE, ETS, U. Quebec, Canada, GUILLAUME BEARDSELL, GUY DUMAS, U. Laval, Canada — In this work, we use DNS to study the reconnection of two orthogonal vortices following the approach of Boratav et al. (1992). For equal circulation vortices we observe the classical reconnection process (Hussain & Duraisamy, 2011). Our main interest though is on the interaction of unequal strength vortices for which only partial reconnection can occur. Typically in these latter cases, the weak vortex (Γ_2) is seen to deform and wrap itself around the strong one (Γ_1) to (partially) reconnect in an anti-parallel configuration similar to what is observed in Marshall et al. (2001). Each branch of the broken weak vortex then forms a spiral structure around the main one; the weaker the vortex, the stronger the spiral. This results in two "circulation jumps" on the main vortex that propagate away from each other, leaving behind them an altered main vortex with reduced circulation. For Reynolds numbers (Γ_1/ν) in the order of 10^3 and circulation ratios $0.1 \leq \Gamma_2/\Gamma_1 \leq 0.9$, we look at the internal struture of the main vortex with a particular attention to the propagating vorticity structures. These structures are very similar to what has been previously observed in the evolution of four-vortex systems (Dufresne & Winckelmans, 2005).

[1]Funding from NSERC Discovery Grant and Scholarship programs is acknowledged by the authors

9:18AM M12.00007 Phase space pattern formation: the single-wave model[1] , P.J. MORRISON, University of Texas at Austin, N.J. BALMFORTH, University of British Columbia, J.-L. THIFFEAULT, University of Wisconsin — Pattern formation in physical systems has received considerable attention, much of which is based on Ginzburg–Landau type systems with advective and diffusive nonlinearity and dispersion. In contrast, the single-wave model (SWM), a Hamiltonian mean-field model, arises in many physical contexts that share common pattern forming behavior. Although the SWM was originally derived in nonlinear plasma theory, where it describes the behavior near threshold and subsequent nonlinear evolution of unstable plasma waves, it arises in fluid mechanics, specifically vortex dyanmics, and also applies to galactic dynamics, the XY and Potts models of condensed matter physics, and general long-range Hamiltonian mean field models. The SWM is a normal form equation for systems that transition to instability with modes emerging from a continuous spectrum (critical layers) and it describes there subsequent nonlinear behavior and pattern formation. This talk surveys SWM phenomena as described in a recent review article.[2]

[1]PJM was supported by U.S. Dept. of Energy Contract # DE-FG05-80ET-53088.

[2]N. Balmforth, P. J. Morrison, and J-L. Thiffeault, Reviews of Modern Physics, to appear (2012).

9:31AM M12.00008 Creation and Dynamics of Knotted Vortices , DUSTIN KLECKNER, MARTIN SCHEELER, WILLIAM IRVINE, University of Chicago — Fluid vortex loops linked together or tied into knots are the basis of a topological interpretation of fluid mechanics. In perfect fluids, the linking of vortex lines is preserved indefinitely and associated with a conserved quantity known as helicity. The situation is considerably more complicated in real fluids - even superfluids - because the vortex topology can change through local reconnections whose dynamics are not well understood. Previous attempts to study these phenomena in experiments have failed because no controlled method existed for making vortex knots in the laboratory. We will describe a method we recently developed for making knotted and linked vortices using 3D-printed hydrofoils. We measure the subsequent evolution of the vortex structures using high-speed laser scanning tomography. We observe that they spontaneously untie/unlink themselves through a series of local reconnections, which we resolve in detail.

9:44AM M12.00009 The Formation Number of Accelerating and Variable Diameter Jet Flows, and a Review of Pinch-Off Criteria[1] , MIKE KRIEG, KAMRAN MOHSENI, University of Florida — This study analyzes vortex ring formation from starting jets with variable jet velocity and diameter and the underlying mechanisms of separation from the feeding shear flow. We assume that the conditions necessary for a vortex ring to separate from the driving shear flow can be identified by a relationship between characteristic velocities of the jet and the vortex ring along the axis of symmetry, and examine multiple pinch-off criteria. A wide variety of jet driving conditions are examined to validate the relationship between pinch-off and characteristic velocities under different constraints, including nozzle type (inclusion of converging radial velocity), acceleration of the jet velocity, and dynamic contraction/expansion of the shear layer diameter. All of these parameters are examined and adjusted independently of each other so that the effect of each jetting parameter can be observed without being affected by the other parameters. Accelerating the jet velocity to compensate for the growing vortex ring substantially increased the formation number of both parallel and converging jet flows. Different definitions of formation time (different time scaling) are also investigated as they pertain to the final vortex ring configuration and the physics of vortex ring formation. It is observed that new definitions of formation time result in jet formation numbers more closely aligned with previous results, suggesting that the new definition corresponds to the physics of vortex ring formation.

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

Tuesday, November 20, 2012 8:00AM - 10:10AM –

Session M13 Richtmyer-Meshkov Instability I 27A - Chair: Snezha Abarzhi, University of Chicago

8:00AM M13.00001 Richtmyer-Meshkov growth of a granular layer , STUART DALZIEL, DAMTP, University of Cambridge — This paper explores the mechanism responsible for the growth of Richtmyer-Meshkov instability in the novel context of the impulsive acceleration of a granular layer. With the classical instability, when a density interface is impulsively accelerating towards the medium with a lower density, the phase of interfacial perturbations is reversed and the deposited kinetic energy leads to growth of the perturbations. The higher density medium forms "spikes" as it penetrates into the lower density medium, while the lower density medium forms "bubbles" as it penetrates into the denser medium. Simple laboratory experiments are used to demonstrate that this Richtmyer-Meshkov growth mechanism can act on the surface of a granular layer, forming structures reminiscent of the classical case. This happens despite the fundamental differences in the way stresses are communicated.

8:13AM M13.00002 Richtmyer-Meshkov instability induced by strong shocks[1] , MILOS STANIC, University of Alabama, Huntsville, AL, USA, ROBERT STELLINGVERF, Srellingverf Consulting, Huntsville, AL, USA, JASON CASSIBRY, University of Alabama, Huntsville, USA, SNEZHANA ABARZHI, University of Chicago, Chicago, IL, USA — We systematically study the Richtmyer-Meshkov instability (RMI) induced by strong shocks for fluids with contrasting densities and with small and large amplitude initial perturbations imposed at the fluid interface. The Smoothed particle hydrodynamics code (SPHC) is employed to ensure accurate shock capturing, interface tracking, and accounting for the dissipation processes. Simulations results achieve good agreement with existing experiments and with the theoretical analyses including zero-order theory describing the post-shock background motion of the fluids, linear theory providing RMI growth-rate in a broad range of the Mach and Atwood numbers, weakly nonlinear theory accounting for the effect of the initial perturbation amplitude on RMI growth-rate, and highly nonlinear theory describing evolution of RM bubble front. We find that for strong-shock-driven RMI the background motion is supersonic, and the interfacial mixing can be sub-sonic or supersonic. Significant part of the shock energy goes into compression and background motion of the fluids, and only a small portion remains for interfacial mixing. The initial perturbation amplitude appears a key factor of RMI evolution. It strongly influences the dynamics of the interface, in the fluid bulk, and the transmitted shock.

[1]Work is supported by NSF, award 1004330

8:26AM M13.00003 Scale coupling in Richtmyer-Meshkov flows induced by strong shocks[1] , R.F. STELLINGVERF, Stellingverf Consulting, Huntsville, AL, USA, M. STANIC, J.T. CASSIBRY, University of Alabama, Huntsville, USA, S.I. ABARZHI, University of Chicago, Chicago, IL, USA — We report an integrated study of the Richtmyer-Meshkov (RM) flows under high energy density relevant conditions by means of smoothed particle hydrodynamics simulations and theoretical analysis. We show that significant amount of the shock energy goes into the compression and background of the fluids (that is supersonic), and only a small portion remains for interfacial mixing (that can be sub-sonic or supersonic). At late times, the RM bubbles flatten and decelerate, specific drag force decays quickly, the interfacial motion tends to be inertial, and the flow remains laminar rather than turbulent. At early times, shear-driven Kelvin-Helmholtz structures appear at the interface. At late times the velocity field is non-uniform and is characterized by intense dynamics in a vicinity of the front, effectively no motion in the bulk (rather than the background motion), and the checkerboard velocity patterns, which are induced by reverse cumulative jets. These jets appear in the fluid bulk and are accompanied by hot spots - local heterogeneous microstructures with temperature substantially higher than that in the ambient. Our results show that RMI dynamics is a multi-scale and heterogeneous process with a complicated character of scale coupling at the interface and in the bulk.

[1]Work is supported by NSF, award 1004330.

8:39AM M13.00004 Three-dimensional modeling of Richtmyer-Meshkov instability[1] , MICHAEL ANDERSON, Illinoisrocstar LLC, PETER VOROBIEFF, C. RANDALL TRUMAN, The University of New Mexico, SANJAY KUMAR, University of Texas - Brownsville — We explore the use of CFD to accurately model the three-dimensional structure of Richtmyer-Meshkov instability (RMI), matching numerical data with new experimental results. Earlier experimental work focused on visualizing planar slices of the initial conditions to observe the formation and growth of the RMI. It was often assumed in the past that the initial conditions are relatively constant in the third dimension and any variation that existed had little effect on the resulting instabilities. Recent experiments have provided quantitative data revealing the three-dimensional structure of the flow. Two computational tools were used in this work, the commerical CFD code FLUENT and the Second-order Hydrodynamic Automatic Mesh Refinement Code (SHAMRC). To reproduce the experimental results, it was necessary to faithfully reproduce the initial conditions on a gaseous density interface prior to shock arrival. This was achieved with FLUENT. Then SHAMRC was used to model the shock interaction and subsequent formation of the RMI. Results are presented for 2D and 3D planar shock/heavy gas column interaction and explore the formation of structures observed experimentally.

[1]This research is supported by NNSA.

8:52AM M13.00005 Integrated study of non-uniform structures in Richtmyer-Meshkov unstable flows by means of theoretical analysis, Lagrangian and Eulerian numerical simulations, and experiments[1] , J.T. CASSIBRY, M. STANIC, University of Alabama, Huntsville, AL, USA, R.F. STELLINGVERF, Stellingverf Consulting, Huntsville, AL, USA, J. MCFARLAND, D. RANJAN, University of Texas A&M, TX, USA, R. BONAZZA, University of Wisconsin, Madison, WI, USA, S.I. ABARZHI, University of Chicago, Chicago, IL, USA — We conducted the integrated study of the Richtmyer-Meshkov flow by means of theoretical analysis, Lagrangian and Eulerian numerical simulations, and experiments achieving good qualitative and quantitative agreement. In our study, Mach numbers are moderate, Atwood numbers are high, initial perturbation amplitudes are finite, and the initial perturbation is coherent. We showed that in this regime, the velocity at which the interface would move if it would be ideally planar is a relevant parameter, as it tracks the amount of momentum and energy deposited by shock at the interface. The amplitude of the initial perturbation is one of key factors of RMI evolution. In case of large amplitudes, the vector and scalar fields in the fluid bulk are non-uniform. The flow non-uniformities include cumulative reverse jets, checkerboards velocity pattern, shock-focusing effects, and local hot spots with temperature substantially higher than that in the ambient. The dynamics of the nonlinear flow is shown to have an essentially multi-scale character.

[1]Work is supported by NSF, award 1004330.

9:05AM M13.00006 Numerical simulations of cylindrical Richtmyer-Meshkov instability at a solid-gas interface , A. LÓPEZ ORTEGA, M. LOMBARDINI, P.T. BARTON, D.I. PULLIN, D.I. MEIRON, California Institute of Technology — Richtmyer–Meshkov flows occur in a wide range of physical phenomena and are of particular interest in shock compression of condensed matter. In this presentation, we discuss numerical simulations of a perturbed, solid–gas interface following the passage of a shock wave in cylindrical geometries. Results are obtained using a shock-capturing scheme applied to the equations of motion for contiguous gaseous and elastic–plastic solid media in a level set-based, multi–material and fully compressible Eulerian framework. Multiple Atwood ratios, initial amplitudes and shock strengths are investigated. Results show that fluid–solid interfaces become unstable when a plasticity model is added to the description of the solid. Under certain initial conditions, ejecta can be formed. This contrasts to previous results (López Ortega et. al, *PRE*, 2010) for purely elastic solids, in which the interface exhibited stable behavior.

9:18AM M13.00007 Shock-accelerated gas cylinder: a Mach number study[1] , TENNILLE BERNARD, PATRICK WAYNE, CLINT CORBIN, C. RANDALL TRUMAN, PETER VOROBIEFF, The University of New Mexico, SANJAY KUMAR, University of Texas - Brownsville, MICHAEL ANDERSON, Illinoisrocstar LLC — We present an experimental study of the evolution of Richtmyer-Meshkov instability and secondary instabilities at a nominally cylindrical density interface under the influence of a planar shock wave traveling at Mach numbers from 1.2 to 2.4. Shock acceleration of the heavy gas (SF_6) cylinder creates not only the expected primary instability resulting in the formation of a pair of counter-rotating vortex columns, but also produces a prominent spike-like feature. Secondary instabilities (*e.g.,* shear-driven) then develop in the spike. The spike formation most likely occurs due to shock focusing as the shock passes through the initial conditions. It is noteworthy that secondary instabilities in the spike were first observed numerically, and then their existence was confirmed experimentally using laser-induced fluorescence.

[1]This research is supported by NNSA.

9:31AM M13.00008 Characteristics of Richtmyer Meshkov Instability in a Spherical Geometry[1] , ANTHONY NELSON, PRAVEEN RAMAPRABHU, University of North Carolina at Charlotte, UNCC TEAM — We describe recent numerical simulations of the single-mode Richtmyer-Meshkov (RM) instability in a spherical geometry. The simulations were performed using the astrophysical FLASH code in two-dimensions in spherical coordinates. Two kinds of RM problems were setup to exploit the effect of shock convergence on perturbation growth. The first set of simulations had low Atwood number interfaces with large perturbations, subject to a Mach 1.2 shock. This set was established to investigate the result of direct contact between the interface and the converging/strengthening shock wave. Secondly, we investigated high Atwood number interfaces with high wavenumber perturbations, subject to a Mach 6 shock. For these simulations, we studied the interaction between the contact discontinuity and a strong converging shock when in close proximity. We expect the single-mode results to inform multimode growth relevant to applications.

[1]We acknowledge funding from the NC Space Grant Consortium.

9:44AM M13.00009 RANS Initialization and Validation in Shock-Driven Turbulent Mixing , FERNANDO GRINSTEIN, BRIAN HAINES, JOHN SCHWARZKOPF, Los Alamos National Laboratory — We investigate a working framework for testing unsteady engineering model initialization and closures based on comparing moments extracted from ensemble-averaged 3D LES data and those predicted directly by a 2D, variable-density, compressible, RANS model. The particular focus is shock-driven turbulent material mixing, and the prototypical case considered is the inverse chevron shock-tube configuration for which laboratory and LES studies have been previously reported. LES results are validated through comparison with previous LES and available experimental data; sensitivity to initial material interface conditions, grid resolution, model, and closure specifics are examined.

9:57AM M13.00010 Reacting H2-O2 Richtmyer-Meshkov Instability Simulations Using Detailed Chemistry[1] , PRAVEEN RAMAPRABHU, NITESH ATTAL, University of North Carolina at Charlotte, SUKESH ROY, Spectral Energies LLC, JAMES GORD, Air Force Research Laboratory — Interaction of a shockwave with a flame enhances supersonic mixing and detonation, and is of importance to the design of supersonic combustors, internal combustion engines and fire safety. The Richtmyer-Meshkov instability plays a significant role in these phenomena. We present numerical results of a reacting Richtmyer-Meshkov instability (RMI) triggered by the interaction of a shock with a sinusoidally perturbed H2-O2 diffusion flame. The simulations were performed using a modified version of the astrophysical FLASH code [1]. A detailed H2-O2 reaction mechanism [2] coupled with an operator-split, 2nd order PPM method in FLASH was used to investigate the effect of RMI induced mixing on the flame. A parametric study for shock Mach numbers ranging from 1.2-3 over Atwood numbers of 0.4-0.65 was carried out, and the results will be presented. The detailed flame dynamics upon reshock will also be discussed.

[1] B. Fryxell, K. Olson, P. Ricker, F. Timmes, M. Zingale, D. Lamb, P. MacNeice, R. Rosner, J. Truran and H. Tufo, Astrophys. J., Suppl. Ser. 131, 273 (2000).
[2] G. Billet, J. Comput. Phys. 204, 319 (2005).

[1]We acknowledge funding from Spectral Energies LLC.

Tuesday, November 20, 2012 8:00AM - 9:57AM –

Session M14 Rotating Flows II: Taylor-Couette Flow 27B - Chair: Patrice Meunier, Aix Marseille University

8:00AM M14.00001 Experimental study of the statistics of a gravity-wave instability in a Taylor-Couette system with free surface , JULIAN MARTINEZ MERCADO, CRISTOBAL ARRATIA, NICOLAS MUJICA, Departamento de Fisica, FCFM, Universidad de Chile — In this work, we study the occurrence of a gravity-wave instability in a turbulent Taylor-Couette system with a free surface. In such configuration the system can bifurcate from an axisymmetric turbulent base state to a $m = 1$ gravity wave state, where a wave grows from a resonant mode of the free surface taking the energy from the turbulent base state. We use the Froude number $F_r = (a\omega)^2/gh$ to characterize the bifurcation, where a is the radius of the inner cylinder, ω its angular velocity, g the gravity acceleration and h the height of the free surface. We show that the observed instability is subcritical, presenting bistability and hysteresis. The measured bifurcation curve can be fitted with the deterministic amplitude equation $\partial_t u = \epsilon u + \nu u^2 - \gamma u^3$, being u the wave's amplitude, although differences are observed due to noise induced by turbulence. The growing rate of the wave's amplitude σ varies linearly with $F_r - F_{rc}$. Moreover, the probability distribution of the wave's amplitude can be expressed as a functional of the form $\ln c_0 - c_1 u^2 + c_2 u^3 - c_3 u^4$, resulting from the use of a Fokker-Planck equation to obtain the probability distribution for this type of bifurcation.

8:13AM M14.00002 Transitions and Reynolds number scaling in quasi-Keplerian Taylor-Couette flow , HANSEN NORDSIEK, DANIEL LATHROP, University of Maryland at College Park — Experimental investigations of the Reynolds number dependence of the torque and wallshear stress for Taylor-Couette flow in the quasi-Keplerian regime (Rayleigh stable anticyclonic flow) are presented in the range of $300 < Re < 10^5$. The Taylor-Couette experiment has independently rotating inner and outer cylinders, a radius ratio of 0.724, an aspect ratio of 11.42, and axial boundaries that rotate with the outer cylinder. The torque required to rotate the inner cylinder at a constant angular velocity, and the wall shear stress at the outer boundary are precisely measured as a function of the Reynolds number for several values of the Rossby number, which compares shear to global rotation. We compare our measurements with previous experiments and simulations, and discuss potential implications for the hydrodynamic contribution to angular momentum transport in astrophysical flows.

8:26AM M14.00003 Reynolds Number Effects on Turbulent Characteristics of Taylor-Couette Flow

, JOONHWI PARK, NAOYA FUKUSHIMA, MASAYASU SHIMURA, MAMORU TANAHASHI, TOSHIO MIYAUCHI, Tokyo Institute of Technology — Laminar and turbulent Taylor-Couette flow is of great importance in a wide range of engineering applications, such as viscosity measurement devices, rotating machineries and reactors. In this study, we focus on turbulent Taylor-Couette flow with a fixed outer cylinder and a rotating inner cylinder. Direct numerical simulation (DNS) of turbulent Taylor-Couette flow has been conducted to investigate turbulent characteristics including Reynolds stress budget at Reynolds number from 8000 to 20000. Reynolds number, Re, is defined by gap width and rotating speed of inner cylinder. In this range of Re, turbulent characteristics are expected to change around Re=10000, referring to Wendt's empirical formula. Averaged torque from DNS agrees well with Wendt's empirical formula and torque transition is confirmed around Re=10000. Averaged azimuthal velocity normalized by friction velocity on inner/outer wall increases in logarithmic region with increase in Re. All components of Reynolds stress tensor also increase in all domain. The minute movement of center of Taylor vortices is observed spatially and temporally when Re is over 12000. Finally, Reynolds stress budgets are investigated to figure out Reynolds number effects on turbulent statistics in detail.

8:39AM M14.00004 New considerations for centrifugal buoyancy effects in fast rotating flows

, JOSE MANUEL LOPEZ ALONSO, FRANCISCO MARQUES, Departament de fisica aplicada, Universitat Politecnica de Catalunya, MARC AVILA, Friedrich-Alexander-Universitat Erlangen-Nurnberg, Cauerstrasse 4, 91058 Erlangen, Germany — Boussinesq type approximations accounting for centrifugal buoyancy are well-known and have been used with remarkable results in problems where a distinguised frame of reference is readily identified. However, it does not consider those flows where different parts may rotate independently, such as Taylor-Couette flows with stratification and/or heating, cylindrical containers with the lids rotating at different angular velocities,etc... In these flows, there is not a unique angular velocity and the differences among them may originate that terms which are not considered by the classical Boussinesq approximation become important. Moreover, this centrifugal effect is not only important when we have rotating walls, but also if a strong vortex appears dynamically in the interior of the domain. We propose a new and easy way to introduce the centrifugal buoyancy into the Navier-Stokes equations which takes into account the previous considerations. We present a linear analysis of stability in an axially periodic Taylor-Couette system subjected to a negative gradient of temperature in order to illustrate the differences of using both approximations when considering the centrifugal effects.

8:52AM M14.00005 Symmetry-breaking Hopf bifurcations to 1-, 2-, and 3-tori in small-aspect-ratio counter-rotating Taylor-Couette flow

, SEBASTIAN ALTMEYER, YOUNGHAE DO, Department of Mathematics, Kyungpook National University, FRANCISCO MARQUES, Department of Fisca Aplicada, Universitat Politecnica de Catalanya, JUAN M. LOPEZ, School of Mathematicl and Statistical Sciences, Arizona State University — Taylor-Couette flow in a small aspect-ratio wide-gap annulus in the counter-rotating regime is investigated by solving the full 3D Navier-Stokes equations. The system is invariant under rotations about the axis, reflection about the mid-plane, and time translations. A systematic investigation is presented, both in terms of the flow physics, the numerical simulations and from a dynamical systems perspective. The dynamics are primarily associated with the behavior of the jet of angular momentum that emerges from the inner cylinder boundary layer at about the mid-plane. The sequence of bifurcations as the rotation is increased consists of a Hopf bifurcation breaking the reflection symmetry and leading to an axisymmetric limit cycle associated to an invariant one-torus manifold with a spatio-temporal symmetry. This undergoes a Hopf bifurcation breaking axisymmetry, leading to quasi-periodic solutions evolving on a 2-torus that is only setwise symmetric due to precesion. These undergo a further Hopf bifurcation introducing a third incommensurate frequency leading to a 3-torus. On it,as the rotation is further increased, a SNIC bifurcation happens, destroying the 3-torus and leaving a pair of symmetrically related 2-tori states on which all symmetries of the system have been broken.

9:05AM M14.00006 Influence of Rotation Number to Coherent Structures and Torque Scaling in Turbulent Taylor-Couette Flow[1]

, SEDAT TOKGOZ, LaVision GmbH/TU Delft, GERRIT ELSINGA, RENE DELFOS, JERRY WESTERWEEL, TU Delft — Flow between two coaxial cylinders is called Taylor-Couette flow and has been studied extensively over the years. Due to the closed and well controlled character of the system, experimental studies with different measurement techniques mostly focused on the turbulent flow. Torque measurements performed at the turbulent range of Reynolds numbers showed change of the torque scaling with relative rotation speeds of the cylinders (i.e. rotation number). In this study, we use tomographic PIV to capture instantaneous three-dimensional flow structures in turbulent Taylor-Couette flow in order to study the mechanism that is responsible for the change of the torque. Time-averaging and auto-correlation of the data confirm the change of coherent structures with the rotation number in the mean flow. Spatial filtering of the instantaneous vector fields enables separating the contributions of small and large scale motions to the Reynolds stress. We show that combination of large scale azimuthal-small scale radial and large scale azimuthal-large scale radial motions are the dominant ones that are effecting the torque. Additionally, we observe change of the organisation of the coherent large scale structures with the rotation number, in relation to the change of the torque scaling.

[1]This research was supported by FP7 Marie Curie Initial Training Network, ITN 215723, "Surface Physics for Advanced Manufacturing" project.

9:18AM M14.00007 Experimental studies of turbulence lifetimes in differentially rotating flows

, E.M. EDLUND, Z. YAN, E.J. SPENCE, A.H. ROACH, J. RHOADS, H. JI, Princeton Plasma Physics Laboratory — Inference of accretion rates from observations of stellar systems suggests inward mass fluxes which can only be reasonably explained by a turbulent transport process. While the magneto-rotational instability (MRI) is likely active in systems above a critical ionization, there remains some question as to whether the MRI can be active in cooler bodies such as proto-planetary systems, and if not, what mechanism is then responsible for angular momentum transport? Keplerian rotation profiles are hydrodynamically linearly stable in the inviscid limit, however, it is not known if there exists a subcritical transition. A series of studies in the Hydrodynamic Turbulence Experiment (HTX), a modified Taylor-Couette device, have explored quiescent flows in the quasi-Keplerian regime. Operating in the wide-gap limit and with split axial boundaries to control the Ekman circulation, azimuthal flows in HTX can be brought very close to ideal Couette. These flows are subjected to external perturbations to test their ability to sustain incompressible hydrodynamic turbulence. Under no circumstances has a subcritical transition to turbulence been observed. Turbulence decay lifetimes are measured and compared to theoretical models.

9:31AM M14.00008 Bifurcation, thin film structure and collapse in Newton's bucket[1]

, SHOMEEK MUKHOPADHYAY, University of California, Riverside, JOSHUA DIJKSMAN, Physics Department, Duke University, TOM WITELSKI, Mathematics Department, Duke University, RICHARD MCLAUGHLIN, ROBERTO CAMASSA, Mathematics Department,University of North Carolina- Chapel Hill, BOB BEHRINGER, Physics Department, Duke University — The understanding of rotating thin film flows is of great fundamental and practical (spin coating, geophysical flows) importance. In this talk we will present our ongoing work with the second generation of a spin coating apparatus that we call "Newton's bucket," extending on previous work [Mukhopadhyay and Behringer (J. Phys, 2009) and Mukhopadhyay et. al. (Phys. Rev. E, 2011)]. We study the bifurcation of the 'non-classical' dry spot that develops above a critical rotation rate. We observe a nontrivial fine structure in the contact line that connects the dry spot with the fluid reservoir and measure the collapse dynamics of the fluid reservoir by means of high speed imaging. We compare our observations to numerical solutions of the lubrication approximation.

[1]Supported by NSF-DMS-09-68252.

9:44AM M14.00009 Stability of solution branches in infinite rotating disk flow[1] , KEVIN VAN EETEN, JOHN VAN DER SCHAAF, GERT-JAN VAN HEIJST, JAAP SCHOUTEN, Eindhoven University of Technology — The stability of steady solutions of the Navier-Stokes equations for the problem of viscous flow between an infinite rotating disk and an infinite stationary disk is investigated. A random disturbance is applied to five velocity profiles at $t = 0$, after which the disturbance propagation, $\Delta(t)$, defined as the squared difference of the azimuthal velocity at time t with the steady state azimuthal velocity, is determined. From this propagation data, the Lyapunov exponents are obtained as a function of the Reynolds number. It was found that four of the five solution branches (including the Batchelor solution) are Lyapunov stable. The Stewartson solution, on the other hand, was found to have a positive Lyapunov exponent and diverged from its initial state to a Batchelor type of flow. The mechanism with which the non-viscous core obtains its angular momentum during this transition was identified as being dominated by radial convection from larger radii towards the axis of rotation.

[1]The European Research Council (ERC) is gratefully acknowledged for their financial support to this project (ERC Contract No. 227010)

Tuesday, November 20, 2012 8:00AM - 10:10AM –

Session M15 Biofluids: Microswimmer Suspensions 28A - Chair: Saverio Spagnolie, University of Wisconsin

8:00AM M15.00001 Concentrated active suspensions: Kinetic theory, linear stability and numerical simulations , BARATH EZHILAN, Department of Mechanical Science and Engineering, University of Illinois at Urbana-Champaign, Urbana, Illinois 61801, USA, MICHAEL SHELLEY, Courant Institute of Mathematical Sciences, New York University, New York, New York 10012, USA, DAVID SAINTILLAN, Department of Mechanical Science and Engineering, University of Illinois at Urbana-Champaign, Urbana, Illinois 61801, USA — We study concentrated suspensions of self-propelled rod-like particles using a kinetic model which accounts for local hydrodynamic and steric interactions. We report a base state transition from an isotropic to a nematic orientation distribution beyond a critical effective volume fraction consistent with the Doi-Edwards theory for passive rod-like particles (Doi and Edwards 1986). We analyze the kinetic model linearized near the isotropic and nematic basestates and show that steric interactions have a destabilizing effect causing both pusher and puller suspensions to be subject to instabilities. These predictions from the linear theory are confirmed using fully nonlinear three-dimensional numerical simulations of the kinetic equations, which also demonstrate large-scale fluctuations of number density and nematic order parameter.

8:13AM M15.00002 Simulation and continuum modelling of a non-uniform suspension of spherical squirmers , TIMOTHY PEDLEY, University of Cambridge, TAKUJI ISHIKAWA, Tohoku University, Sendai, Japan — Stokesian dynamics simulations are performed for a non-dilute suspension of identical spherical squirmers (cells) whose initial concentration distribution $c(x,t)$ is sinusoidal in x. It is found that the c-distribution overshoots its mean, so that there are times at which the maximum values of c occur at locations where initially c was a minimum and vice versa. This is not consistent with a purely diffusive model. We consider continuum models in terms of the cell conservation equation, incorporating the average cell swimming velocity $\boldsymbol{U}$ and representing random cell motion (resulting solely from hydrodynamic interaction between cells) by a diffusivity tensor $\mathbf{D}$. If the values of $\boldsymbol{U}$ and $\mathbf{D}$ obtained from the simulation are used in the equations, the results agree well with the simulations. However, if we start from the Fokker-Planck equation for the pdf of orientation, representing hydrodynamic interactions by a constant rotational diffusivity, and truncating the sequence of moment equations at the first or second moment, agreement is not very good. We discuss what would be needed in a continuum model for it to be able to predict $\boldsymbol{U}$ and $\mathbf{D}$ accurately, without doing the full simulation first.

8:26AM M15.00003 Out-of-Equilibriumness of Light Activated Colloids , JEREMIE PALACCI, CSMR, N, STEFANO SACANNA, CSMR, NYU, USA, ASHER PRESKA-STEINBERG, Brandeis University, USA, DAVID PINE, PAUL CHAIKIN, CSMR, NYU, USA — Self-propelled micro-particles are intrinsically out-of-equilibrium. This renders their physics far richer than that of passive colloids while relaxing some thermodynamical constraints and give rise to the emergence of complex phenomena e.g. collective behavior, swarming... We will present a new form of self-assembly originating from non-equilibrium driving forces. When activated by light, a set of new self-propelled particles spontaneously assemble into living crystals which behaves as "self-propelled colloidal carpets" steerable with an external magnetic field. We will show that this phenomenon is intrinsically out-of-equilibrium and originates in the competition between self-propulsion, particles collisions and attractive interactions. We will also present present surprising behaviors of these particles in confined environments.

8:39AM M15.00004 Extensive active suspension , ANSHENG JHANG, MICHEAL SHELLEY, NYU — A suspension of rod-like growing particles, like a suspension of self-propelled particles, can exhibit complex dynamics as a result of long-ranged hydrodynamic interactions. Such suspensions can occur in bacterial colonies, liquid crystals phase transitions, or micro-tubules with kinesin. As they grow, they exert stress on the fluid which is similar to the case of swimming pushers. We use a kinetic model to study the dilute limit case. We will discuss the cases in terms of domain shapes like periodic boundary domains, simply connected domain, and annulus like domain.

8:52AM M15.00005 Flow of active suspensions and biased swimming[1] , SALIMA RAFAI, PHILIPPE PEYLA, XABEL GARCIA, GUNTARS KITENBERGS, MICHAËL GARCIA, CNRS/Grenoble University, LIPHY TEAM — It is a challenge to understand the hydrodynamics associated with individual or collective motion of microswimmers through their fluid-mediated interactions in order for instance to manipulate the cells efficiently for some applications purposes. The motion of these micro-organisms can be often affected by the presence of gradients leading to a biased random walk (chemotaxis in the presence of chemicals, gyrotaxis in a gravity field, phototaxis under light exposure). In this study, we present our experimental results concerning the coupling of a Poiseuille flow with the biased random walk of Chlamydomonas Reinhardtii, a green unicellular micro-alga. This is done by illuminating the microswimmer suspension while flowing in a microchannel device. We show that one can obtain a spontaneous and reversible migration and separation of the microalgae suspension from the rest of the suspending medium under illumination and then dynamically control the concentration of the suspension with light. We present a simple model that accounts for the observed phenomenon.

[1]We thank the ANR MOSICOB and MICMACSWIM.

9:05AM M15.00006 Spontaneous Circulation of Confined Active Suspensions , FRANCIS WOODHOUSE, RAYMOND GOLDSTEIN, DAMTP, University of Cambridge — Many active fluid systems encountered in biology are set in total geometric confinement; cytoplasmic streaming is a prominent and ubiquitous example. Using the simple paradigm of a dilute dipolar swimmer suspension, we demonstrate that the two key constraints of circular confinement and fluid incompressibility yield qualitatively new dynamics, effectively quantizing the behaviour regimes. We show analytically that there is an activity threshold for spontaneous auto-circulation and verify this numerically. Long-time non-linear behaviour is investigated via simulations, which reveal steady states displaying nematic defect separation and a high-activity bifurcation to an oscillatory regime.

9:18AM M15.00007 Simulations of bacterial chemotaxis in the turbulent ocean , ROMAIN WATTEAUX, JOHN TAYLOR, Department of Applied Mathematics and Theoretical Physics, University of Cambridge — Nearly half of the global primary production occurs in the oceans. Between 30 and 50% of the newly generated carbon is released into the surrounding water as dissolved organic matter (DOM), and is almost exclusively accessible to bacteria. By consuming DOM and returning the carbon to the marine food web, bacteria act as recyclers. Some bacteria are motile and have the ability to modify their otherwise random motility in response to a chemical cue, a process known as chemotaxis. It has been recently shown that motile bacteria can benefit from turbulence by clustering around thin DOM filaments, thereby increasing their uptake (Taylor and Stocker, Science, *accepted*). Here, we extend the previous analysis by considering weakly diffusive DOM patches (with a Schmidt number, $Sc = \nu/\kappa_C$ up to 1000), and examine the counteracting effects of chemotaxis and random motility. Using direct numerical simulations (DNS), we find that the uptake enhancement depends on characteristic length scales of DOM and bacteria filaments, which in turn depend on three parameters: the turbulent dissipation rate, the bacteria swimming speed, and the DOM diffusivity. By exploring a range of realistic parameter values, we are able to characterize the advantage afforded by motility.

9:31AM M15.00008 Meso-scale turbulence in living fluids[1] , JORN DUNKEL, DAMTP, University of Cambridge, RIK WENSINK, CNRS, Universite Paris-Sud 11, SEBASTIAN HEIDENREICH, Physikalisch Technische Bundesanstalt Berlin, KNUT DRESCHER, Princeton University, RAY GOLDSTEIN, DAMTP, University of Cambridge, HARTMUT LOEWEN, Heinrich Heine Universitat Duesseldorf, JULIA YEOMANS, University of Oxford — The mathematical characterization of turbulence phenomena in active non-equilibrium fluids proves even more difficult than for conventional liquids or gases. It is not known which features of turbulent phases in living matter are universal or system-specific, or which generalizations of the Navier-Stokes equations are able to describe them adequately. We combine experiments, particle simulations, and continuum theory to identify the statistical properties of self-sustained meso-scale turbulence in active systems. To study how dimensionality and boundary conditions affect collective bacterial dynamics, we measured energy spectra and structure functions in dense Bacillus subtilis suspensions in quasi-2D and 3D geometries. Our experimental results for the bacterial flow statistics agree well with predictions from a minimal model for self-propelled rods, suggesting that at high concentrations the collective motion of the bacteria is dominated by short-range interactions. To provide a basis for future theoretical studies, we propose a minimal continuum model for incompressible bacterial flow. A detailed numerical analysis of the 2D case shows that this theory can reproduce many of the experimentally observed features of self-sustained active turbulence.

[1]Supported by the ERC, EPSRC and DFG

9:44AM M15.00009 Discrete population dynamics in flows[1] , PRASAD PERLEKAR, Department of Applied Physics, Eindhoven University of Technology, The Netherlands, ROBERTO BENZI, LUCA BIFERALE, Department of Physics, University of Tor Vergata, Italy, HERMAN CLERCX, Department of Applied Physics, Eindhoven University of Technology, The Netherlands, SIMONE PIGOLOTTI, Dept. de Fisica i Eng. Nuclear, Universitat Politecnica de Catalunya Edif. GAIA, Spain, MOGENS JENSEN, The Neils Bohr Institut, Copenhagen, Denmark, DAVID NELSON, Lyman Laboratory of Physics, Harvard University, Cambridge, USA, FEDERICO TOSCHI, Department of Applied Physics, Eindhoven University of Technology, The Netherlands — Bacteria and plankton populations living in oceans and lakes reproduce and die under the influence of turbulent currents. Fluid transport interacts in a complex way with the dynamics of populations because the typical reproduction time of microorganism is comparable with the time scale of the flows. We review recent results on the population dynamics for off-lattice models. We then investigate the role of chaotic/turbulent flows on the dynamic of populations. The populations are modeled as discrete entities (particles) that reproduce, die, and compete with each other. Furthermore, to mimic various seggregation mechanisms like gyrotaxis, chemotaxis, and/or food variability we associate an inertia with the entities. We show that the presence of advecting flows with same "inertial" entities leads to a dramatic reduction in the population sizes and fixation times. We also discuss the interesting case of species with slightly different inertial properties where a long coexistence of species is possible.

[1]We thank FOM for financial support.

9:57AM M15.00010 Mixing by individual swimmers , DMITRI PUSHKIN, Theoretical Physics, University of Oxford, HENRY SHUM, University of Pittsburgh, JULIA YEOMANS, Theoretical Physics, University of Oxford — Despite their evolutionary and technological importance, different biomixing mechanisms, their effectiveness and universality remain poorly understood. In this talk we focus on the Lagrangian transport of the surrounding fluid by swimmers. Low Re passive tracers advected by swimmers move in loops that are, in general, almost closed. We analyze the reasons for this behavior and, as non-closedness of the loops is a natural requirement for an efficient mixing, propose a classification of possible mechanisms for biogenic mixing. Next, we discuss the universal (common to all swimmers) and the swimmer-dependent features of the resulting tracer displacements and analyze the Darwin drift, the total fluid volume displaced by a swimmer passing from and to infinity. We show that the Darwin drift is finite for force-free swimmers and can be decomposed into a universal and a swimmer-dependent part. We illustrate our consideration with examples for model swimmers and biological data.

Tuesday, November 20, 2012 8:00AM - 10:10AM –

Session M16 Biofluids: Blood Cells 28B - Chair: Prosenjit Bagchi, Rutgers University

8:00AM M16.00001 Orbital drift of capsules and red blood cells[1] , DAN CORDASCO, PROSENJIT BAGCHI, Rutgers, The State University of New Jersey — Experiments using deformable red blood cells (RBC) in shear flow showed that the cells orient their symmetry axis towards the plane of shear (C = ∞ orbit) in a high viscosity medium, and along the vorticity direction (C = 0 orbit) in a low viscosity medium. In contrast, rigid ellipsoids in Stokes flow exhibit degenerate trajectories. The degeneracy can be broken by inertial effects, or, deformability. To explore the orientational drift, we conduct a 3D numerical simulation of prolate and oblate capsules and RBC over a range of capillary number, asphericity, and viscosity ratio. Four types of motion are observed: a stable precessing about C = 0, a stable kayaking about C = ∞, an unstable precessing towards C = 0, and a transition from a kayaking to a drifting precession. A prolate capsule with viscosity ratio of one mostly exhibits a kayaking at low asphericity, but mostly a drifting precession at high asphericity. In contrast, an oblate capsule drifts towards C = ∞. In agreement with published experiments, we find that the RBC orients its symmetry axis to C = 0 at high viscosity ratio, and C = ∞ at low viscosity ratio. We also find that the RBC orientation is dependent on the capillary number, implying the role of deformation.

[1]Funded by NSF

8:13AM M16.00002 Mechanical Response of Red Blood Cells Entering a Constriction: Influence of Oxidative Stress , N.F. ZENG, W.D. RISTENPART, Dept. Chemical Engineering and Materials Science, Univ. California Davis — A key determinant of RBC deformability is the level of oxidative stress, i.e., the imbalance of reactive oxygen species (ROS) associated with many disease states. Previous work has shown that oxidative stress rigidifies RBC membranes, but little is known about the mechanical response of RBCs to oxidative stress under physiological shear conditions. Here we show that oxidative stress significantly alters the dynamic mechanical behavior of RBCs undergoing a sudden increase in shear stress. Using high speed video, we tracked the motion of RBCs entering a narrow constriction in a microfluidic channel. Varied concentrations of hydrogen peroxide, a generator of ROS, were added to the RBCs to induce oxidative stress. We demonstrate that an H2O2 concentration as low as 30μM significantly decreases the percentage of RBCs undergoing stretching and twisting motions, while simultaneously increasing the percentage of RBCs undergoing tumbling motions. A key observation is that the H2O2 treatment reduced the average RBC volume by up to 30%, suggesting that an increase in intracellular viscosity increased the propensity for RBCs to tumble.

8:26AM M16.00003 Skeleton deformation of red blood cells during tank treading motions[1] , QIANG ZHU, UC San Diego, ZHANGLI PENG, M.I.T. — By coupling a fluid-structure interaction algorithm with a three-level multiscale structural model, we simulate the tank treading responses of erythrocytes (red blood cells, or RBC) in shear flows. The fluid motion is depicted within the Stokes-flow framework, and is mathematically formulated with the boundary integral equations. The structural model takes into account the flexible connectivity between the lipid bilayer and the protein skeleton as well as the viscoelastic responses. The concentration of this study is on the transient process involving the development of the local area deformation of the protein skeleton. Under the assumption that the protein skeleton is stress-free in the natural biconcave configuration, our simulations indicate the following properties: (1) During tank treading motions it takes long time for significant area deformations to establish. For cells with diminished connectivity between the lipid bilayer and the protein skeleton (e.g. cells with mutations or defects), the relaxation time will be greatly reduced; (2) Deformations of the skeleton depend on the initial orientation of the cell with respect to the incoming flow; (3) The maximum area expansion occurs around the regions corresponding to the dimples in the original biconcave state; (4) Oscillations in cell geometry (breathing) and orientation (e.g. swinging) are observed.

[1]This work was supported by the National Heart, Lung, and Blood Institute under award number R01HL092793.

8:39AM M16.00004 Deformation of a single red blood cell in bounded Poiseuille flows[1] , LINGLING SHI, TSORNG-WHAY PAN, ROLAND GLOWINSKI, University of Houston — An immersed boundary method (IBM) combined with the elastic spring model is applied to investigate the deformation of a single red blood cell (RBC) in two-dimensional bounded Poiseuille flows. The equilibrium shape of the cell under flow depends on the swelling ratio ((s^*)), the initial angle of the long axis of the cell at the centerline (φ), the maximum velocity of the flow ($u_{\max}$), the membrane bending stiffness of the RBC (k_b), and the height of the microchannel(H). Two motions of oscillation and vacillating breathing of the RBC are observed in narrow channel considered here. The strength of the vacillating-breathing motion depends on degree of confinement and $u_{\max}$. For the different k_b, the RBC obtains the same equilibrium shape for the same capillary number. Parachute shape and bullet-like shape, depending on the angle φ, coexist for the elliptic shape cell with lower $u_{\max}$ in a narrower channel.

[1]NSF Grant No. DMS-0914788

8:52AM M16.00005 On the oscillating motion of a red blood cell in bounded Poiseuille flows[1] , YAO YU, LINGLING SHI, ROLAND GLOWINSKI, TSORNG-WHAY PAN, Department of Mathematics, University of Houston, Houston TX 77204, USA — Two motions of oscillation and vacillating breathing (swing) a RBC have been observed in bounded Poiseuille flows [Phys. Rev. E 85, 16307 (2012)]. To understand such motions, we have studied the motion of a neutrally buoyant rigid particle of the same shape in bounded Poiseuille flows and obtained that the equilibrium height of the mass center, the confined ratio of the long axis of the particle and the channel height, and the initial position of the particle are important factors for having such oscillating motion. But the crucial one is to have the particle interacting with Poiseuille flow with its mass center oscillating about the channel center. When the mass center is always away from the channel center, the particle just keep rotating. Since the mass center of the cell migrates to the channel center in bounded Poiseuille flow in the regime of low Reynolds number, the oscillating motion then is similar to the aforementioned motion as long as the cell keeps the shape of long body.

[1]This work is funded by the US NSF

9:05AM M16.00006 Simulation of the Effect of Red Blood Cell Collisions on Platelet Adsorption , SEAN FITZGIBBON, Stanford Chemical Engineering, HONG ZHAO, Stanford Mechanical Engineering, ERIC SHAQFEH, Stanford Chemical Engineering, Stanford Mechanical Engineering, Stanford Institute for Computational and Mathematical Engineering — The adsorption of platelets to the endothelial wall is an important first step in the clotting process, which is critical to stopping blood loss after trauma. Initial platelet arrest is controlled by very short range interaction between two proteins, von Willibrand Factor and GPIb, so the rate of platelet adsorption is expected to be strongly dependent on the rate at which the platelets sample the wall. With Peclet numbers in the range (10^3 - 10^5), simple diffusive arguments are not sufficient to explain the high rates of platelet adsorption. Using Stokes flow simulations, we show that the platelets' wall sampling rate is significantly increased by interactions with red blood cells. Our simulation models platelets as rigid bodies suspended in a Stokesian linear shear flow. We solve for the flow using standard boundary integral techniques with the appropriate single wall bounded Green's function. Receptor-ligand interactions are represented as Hookean springs with characteristic lifetimes, sizes, and stiffness coefficients. Drag forces are calculated with the reciprocal theorem, and RBC collisions are modelled as AR processes extracted from the large scale suspension simulations of Zhao et al.

9:18AM M16.00007 Microfluidic approach of Sickled Cell Anemia , MANOUK ABKARIAN, ETIENNE LOISEAU, GLADYS MASSIERA, Laboratoire Charles Coulomb, Université Montpellier 2-CNRS — Sickle Cell Anemia is a disorder of the microcirculation caused by a genetic point mutation that produces an altered hemoglobin protein called HbS. HbS self-assembles reversibly into long rope like fibers inside the red blood cells. The resulting distorded sickled red blood cells are believed to block the smallest capillaries of the tissues producing anemia. Despite the large amount of work that provided a thorough understanding of HbS polymerization in bulk as well as in intact red blood cells at rest, no consequent cellular scale approaches of the study of polymerization and its link to the capillary obstruction have been proposed in microflow, although the problem of obstruction is in essence a circulatory problem. Here, we use microfluidic channels, designed to mimic physiological conditions (flow velocity, oxygen concentration, hematocrit...) of the microcirculation to carry out a biomimetic study at the cellular scale of sickled cell vaso-occlusion. We show that flow geometry, oxygen concentration, white blood cells and free hemoglobin S are essential in the formation of original cell aggregates which could play a role in the vaso-occlusion events.

9:31AM M16.00008 Mechanism of vaso-occlusion in sickle cell anemia[1] , HUAN LEI, GEORGE KARNIADAKIS, Brown University — Vaso-occlusion crisis is one of the key hallmark of sickle cell anemia. While early studies suggested that the crisis is caused by blockage of a single elongated cell, recent experimental investigations indicate that vaso-occlusion is a complex process triggered by adhesive interactions among different cell groups in multiple stages. Based on dissipative particle dynamics, a multi-scale model for the sickle red blood cells (SS-RBCs), accounting for diversity in both shapes and cell rigidities, is developed to investigate the mechanism of vaso-occlusion crisis. Using this model, the adhesive dynamics of single SS-RBC was investigated in arterioles. Simulation results indicate that the different cell groups (deformable SS2 RBCs, rigid SS4 RBCs, leukocytes, $etc.$) exhibit heterogeneous adhesive behavior due to the different cell morphologies and membrane rigidities. We further simulate the tube flow of SS-RBC suspensions with different cell fractions. The more adhesive SS2 cells interact with the vascular endothelium and further trap rigid SS4 cells, resulting in vaso-occlusion in vessels less than $15\mu m$. Under inflammation, adherent leukocytes may also trap SS4 cells, resulting in vaso-occlusion in even larger vessels.

[1]This work was supported by the NSF grant CBET-0852948 and the NIH grant R01HL094270.

9:44AM M16.00009 How does confinement affect the dynamics of viscous vesicles and red blood cells? , BADR KAOUI, TIMM KRUGER, JENS HARTING, Eindhoven University of Technology (Eindhoven, The Netherlands) — Despite its significance in microfluidics, the effect of confinement on the transition from the tanktreading (steady motion) to the tumbling (unsteady motion) dynamical state of deformable microparticles has not been studied in detail. In this work, we investigate the dynamics of a single viscous vesicle under confining shear as a general model system for red blood cells, capsules, or viscous droplets. The transition from the tank-treading to the tumbling motion can be triggered by the ratio between internal and external fluid viscosities. Here, we show that the transition can be induced solely by reducing the confinement, keeping the viscosity contrast constant. The observed dynamics results from the variation of the relative importance of viscous-, pressure-, and lubrication-induced torques exerted upon the vesicle. Our findings are of interest for designing future experiments or microfluidic devices: the possibility to trigger the tumbling-to-tank-treading transition either by geometry or viscosity contrast alone opens attractive possibilities for microrheological measurements as well as the detection and diagnosis of diseased red blood cells in confined flow.

9:57AM M16.00010 Desiccation of a pool of blood: from fluid mechanics to forensic investigations[1] , CELINE NICLOUX, Institut de Recherche Criminel de la Gendarmerie Nationale (IRCGN), DAVID BRUTIN, Aix-Marseille University — The evaporation of biological fluids (with droplet configuration) has been studied since a few years due to several applications in medical fields such as medical tests, drug screening, biostabilization... The evaporation of a drop of whole blood leads to the formation of final typical pattern of cracks [1]. Flow motion, adhesion, gelation and fracturation all occur during the evaporation of this complex matter. During the drying, a sol-gel transition develops. The evaporation of a pool of blood is studied in order to link the pattern formation and the evaporation dynamics. We intend to transfer the knowledge acquired for drops on pool to improve the forensic investigations. In this study, we focus on both pool of blood and pure water to determine the transition region from drop to pool and then to characterize the evaporation rate in the pool configuration. The spreading of blood which can be seen as a complex fluid is strongly influenced the substrate nature. The initial contact angle of blood on different substrate nature [2] will influence the maximum thickness of the layer and then will influence the evaporation mass flux.

[1] B. Sobac & D. Brutin, Phys. Rev. E 84, 011603, 2011.
[2] Brutin D., Sobac B., Nicloux C., Journal of Heat Transfer, Vol. 134, 061101, 2012.

[1]The authors gratefully acknowledge the help and the fruitful discussions raised with A. Boccoz.

Tuesday, November 20, 2012 8:00AM - 10:10AM –

Session M17 Biofluids: Microswimmers Experiments II 28C - Chair: Roman Stocker, Massachusetts Institute of Technology

8:00AM M17.00001 Swimming simply: Minimal models and stroke optimization for biological systems[1] , LISA BURTON, Department of Mechanical Engineering, Massachusetts Institute of Technology, JEFFREY S. GUASTO, ROMAN STOCKER, Department of Civil and Environmental Engineering, Massachusetts Institute of Technology, A.E. HOSOI, Department of Mechanical Engineering, Massachusetts Institute of Technology — In this talk, we examine how to represent the kinematics of swimming biological systems. We present a new method of extracting optimal curvature-space basis modes from high-speed video microscopy images of motile spermatozoa by tracking their flagellar kinematics. Using as few as two basis modes to characterize the swimmer's shape, we apply resistive force theory to build a model and predict the swimming speed and net translational and rotational displacement of a sperm cell over any given stroke. This low-order representation of motility yields a complete visualization of the system dynamics. The visualization tools provide refined initialization and intuition for global stroke optimization and improve motion planning by taking advantage of symmetries in the shape space to design a stroke that produces a desired net motion. Comparing the predicted optimal strokes to those observed experimentally enables us to rationalize biological motion by identifying possible optimization goals of the organism. This approach is applicable to a wide array of systems at both low and high Reynolds numbers.

[1]Battelle Memorial Institute and NSF

8:13AM M17.00002 How synthetic microswimmers move, turn, flip, and spread , DAISUKE TAKAGI, ADAM BRAUNSCHWEIG, JUN ZHANG, MICHAEL SHELLEY, New York University — We study the dynamics of bimetallic rods that are propelled catalytically in a fluid of chemical fuel. Our experiments reveal that self-propelled rods near a surface undergo systematic turns, spontaneous flips, and dispersion at a rate that saturates with increasing speed. These phenomena can be explained quantitatively by our mathematical model only by incorporating a slight curvature in the rods. The model shows how minor variations in particle shape lead to major changes in trajectory patterns and diffusivity, and offers insight into dispersion of both synthetic and biological swimmers.

8:26AM M17.00003 Evolved to fail: Bacteria induce flagellar buckling to reorient , KWANGMIN SON, JEFFREY S. GUASTO, ROMAN STOCKER, MIT — Many marine bacteria swim with a single helical flagellum connected to a rotary motor via a 100 nm long universal joint called the "hook." While these bacteria have seemingly just one degree of freedom, allowing them to swim only back and forth, they in fact exhibit large angular reorientations mediated by off-axis "flicks" of their flagellum. High-speed video microscopy revealed the mechanism underpinning this turning behavior: the buckling of the hook during the exceedingly brief (10 ms) forward run that follows a reversal. Direct measurements of the hook's mechanical properties corroborated this result, as the hook's structural stability is governed by the Sperm number, which compares the compressive load from propulsion to the elastic restoring force of the hook. Upon decreasing the Sperm number below a critical value by reducing the swimming speed, the frequency of flicks diminishes sharply, consistent with the criticality of buckling. This elegant, under-actuated turning mechanism appears widespread among marine bacteria and may provide a novel design concept in micro-robotics.

8:39AM M17.00004 Aerotaxis in Bacterial Turbulence , VICENTE FERNANDEZ, ANTOINE BISSON, CINDY BITTON, Massachusetts Institute of Technology, NICOLAS WAISBORD, University of Lyon, STEVEN SMRIGA, ROBERTO RUSCONI, ROMAN STOCKER, Massachusetts Institute of Technology — Concentrated suspensions of motile bacteria exhibit correlated dynamics on spatial scales much larger than an individual bacterium. The resulting flows, visually similar to turbulence, can increase mixing and decrease viscosity. However, it remains unclear to what degree the collective dynamics depend on the motile behavior of bacteria at the individual level. Using a new microfluidic device to create controlled horizontal oxygen gradients, we studied the two dimensional behavior of dense suspensions of *Bacillus subtilis*. This system makes it possible to assess the interplay between the coherent large-scale motions of the suspension, oxygen transport, and the directional response of cells to oxygen gradients (aerotaxis). At the same time, this device has enabled us to examine the onset of bacterial turbulence and its influence on the propagation of the diffusing oxygen front, as the bacteria begin in a dormant state and transition to swimming when exposed to oxygen.

8:52AM M17.00005 Undulatory Swimming in Shear-thinning Fluids[1] , XIAONING SHEN, DAVID GAGNON, PAULO ARRATIA, University of Pennsylvania — Many fluids in which microorganisms move, feed, and reproduce possess shear-rate dependent viscosity behavior (e.g. shear-thinning). Such fluids include wet soil, clay suspension, mucus, and gels. In this talk, we experimentally investigate the effects of shear-rate dependent viscosity on the swimming behavior of the nematode *Caenorhabditis elegans* using velocimetry and tracking methods. Here, aqueous solutions of xanthan gum, which is a rod-like stiff polymer, are used with concentrations varying from the semi-dilute to the concentrated regime. The data is compared to swimming in simple, Newtonian fluids. We find that the nematode swims at an approximately constant speed in the semi-dilute regime. Surprisingly, the nematode exhibits 40% increases in swimming speed once immersed in a concentrated solution. The enhancement in swimming speed seems to be related to the dynamics of rod-like polymer networks formed in concentrated solutions.

[1]This work was supported by NSF-CAREER (CBET)-0954084.

9:05AM M17.00006 Interaction of bacteria and a chemically patterned surface[1] , MARYAM JALALI, MEHDI MOLAEI, JIAN SHENG, Texas Tech University — We are investigating the mechanisms involved in the interactions between bacteria and chemically patched oil-water interface. Using micro-fabrication and soft-lithography, we have engineered a chemically patterned solid surface to mimic the real interfacial environment. Arrays of 2D geometries whose characteristic size ranges from 10μm to 100μm are patterned onto a glass substrate and subsequently functionalized using Octadecyltrichlorosilane (OTS). The photoresist covering geometries is further removed after functionalization. Consequently, a chemically patterned surface with alternating hydrophobic and hydrophilic regions is produced as the substrate for microfluidics. The effects of this surface on bacteria attachment and detachment are evaluated *in-situ*. The growth rates of biofilm are quantified by measuring the morphology of bacterial colony. To elucidate hydrodynamic mechanism involved, bacteria swimming characteristics, such as swimming velocity, angle, tumbling frequency and dispersion, is measured within a microfluidics with a patterned substrate using 3D digital holographic microscopy. Comparative studies on smooth swimming and tumbling capable strains over such surfaces will also be presented.

[1]GoMRI

9:18AM M17.00007 Hydrodynamic behavior of shaking flasks used for producing a recombinant protein by filamentous bacteria , MARIA SOLEDAD CORDOVA AGUILAR, MONICA GARCIA, MAURICIO ALBERTO TRUJILLO-ROLDAN, GABRIEL ASCANIO, ROBERTO ZENIT, ENRIQUE SOTO, Universidad Nacional Autonoma de Mexico — Shake flasks are widely used for culture research. The agitation rate is one of the factors that determines the mass transfer. However, it has not been studied in detail. In this work, a comparison of the hydrodynamic performance for conventional, baffled and coiled spring Erlenmeyer flasks is presented. The velocity fields for a horizontal plane were measured by means of a Particle Image Velocimetry (PIV) technique and high speed videos were recorded to observe the behavior of the interface as a function of the agitation rate. It was observed not only that there is a strong dependence between the geometry and the hydrodynamics, but also there is a good agreement with the results obtained previously by Gamboa et al, in 2011, with the evaluation of the influence of culture conditions of *S. lividans* on protein O-glycosylation. The turbulence intensity increases with shaken rate. However, for the baffled geometry, it was observed a decrease for a critical speed, which is related with the *in*-phase and *out*-phase regions. These results can be an explanation for the variations in protein productivity as a function of the flask geometry and the differences in aggregation morphology and the pattern of O-glycosylation of the recombinant protein.

9:31AM M17.00008 Scalar transport by planktonic swarms[1] , MONICA MARTINEZ-ORTIZ, California Institute of Technology, JOHN O. DABIRI, Graduate Aeronautical Laboratories and Bioengineering, California Institute of Technology — Nutrient and energy transport in the ocean is primarily governed by the action of physical phenomena. In previous studies it has been suggested that aquatic fauna may significantly contribute to this process through the action of the induced drift mechanism. In this investigation, the role of planktonic swarms as ecosystem engineers is assessed through the analysis of scalar transport within a stratified water column. The vertical migration of Artemia salina is controlled via luminescent signals on the top and bottom of the column. The scalar transport of fluorescent dye is visualized and quantified through planar laser induced fluorescence (PLIF). Preliminary results show that the vertical movement of these organisms enhances scalar transport relative to control cases in which only buoyancy forces and diffusion are present.

[1]Funded by the BSF program (2011553)

9:44AM M17.00009 Myco-fluidics: The fluid dynamics of fungal chimerism , MARCUS ROPER, PATRICK HICKEY, UCLA, EMILIE DRESSAIRE, Trinity College, CT, SEBASTIEN ROCH, U. Wisconsin-Madison — Chimeras–fantastical creatures formed as amalgams of many animals–have captured the human imagination since Ancient times. But they are also surprisingly common in Nature. The syncytial cells of filamentous fungi harbor large numbers of nuclei bathed in a single cytoplasm. As a fungus grows these nuclei become genetically diverse, either from mutation or from exchange of nuclei between different fungal individuals, a process that is known to increase the virulence of the fungus and its adaptability. By directly measuring nuclear movement in the model ascomycete fungus *Neurospora crassa*, we show that the fungus' tolerance for internal genetic diversity is enabled by hydrodynamic mixing of nuclei acting at all length scales within the fungal mycelium. Mathematical modeling and experiments in a mutant with altered mycelial morphology reveal some of the exquisite hydraulic engineering necessary to create these mixing flows from spatially coarse pressure gradients.

9:57AM M17.00010 Turbulence from a microorganism's perspective: Does the open ocean feel different than a coral reef? , RACHEL PEPPER, EVAN VARIANO, M.A.R. KOEHL, University of California, Berkeley — Microorganisms in the ocean live in turbulent flows. Swimming microorganisms navigate through the water (e.g. larvae land on suitable substrata, predators find patches of prey), but the mechanisms by which they do so in turbulent flow are poorly understood as are the roles of passive transport versus active behaviors. Because microorganisms are smaller than the Kolmagorov length (the smallest scale of eddies in turbulent flow), they experience turbulence as a series of linear gradients in the velocity that vary in time. While the average strength of these gradients and a timescale can be computed from some typical characteristics of the flow, such as the turbulent kinetic energy or the dissipation rate, there are indications that organisms are disproportionally affected by rare, extreme events. Understanding the frequency of such events in different environments will be critical to understanding how microorganisms respond to and navigate in turbulence. To understand the hydrodynamic cues that microorganisms experience in the ocean we must measure velocity gradients in realistic turbulent flow on the spatial and temporal scales encountered by microorganisms. We have been exploring the effect of the spatial resolution of PIV and DNS of turbulent flow on the presence of velocity gradients of different magnitudes at the scale of microorganisms. Here we present some results of PIV taken at different resolutions in turbulent flow over rough biological substrata to illustrate the challenges of quantifying the fluctuations in velocity gradients encountered by aquatic microorganisms.

Tuesday, November 20, 2012 8:00AM - 10:10AM –

Session M18 Particle Laden Flows VI: Turbulence 28D - Chair: Sarma Rani, University of Alabama in Huntsville

8:00AM M18.00001 On Pair Diffusion and Preferential Concentration of High Stokes Number Particles in Isotropic Turbulence , SARMA RANI, University of Alabama in Huntsville, DONALD KOCH, Cornell University — In this study, we derived the Fokker-Planck equation governing the PDF of pair separation and relative velocity vectors of high St particles. The PDF equation contains a particle-pair diffusion coefficient in relative velocity space. We developed an analytical theory to predict this relative velocity-space pair diffusion coefficient in the limit of high St. Using the diffusion coefficient, Langevin-equation-based stochastic simulations were performed to evolve pair separation and velocity vectors in isotropic turbulence for particle Stokes numbers, $St = 1,\ 2,\ 4,\ 10$, and, 20 and a Taylor micro-scale Reynolds number, $Re_\lambda = 75$. The most significant finding from the Langevin simulations is that our pair diffusivity theory successfully captures the transition of relative velocity PDF from a Gaussian PDF at separations of the order of integral length scale to a non-Gaussian PDF at smaller separations. The pair radial distribution functions (RDFs) computed using our theory show that as the Stokes number increased, particles preferentially accumulate even at integral length scale separations. Another significant finding of our approach is that the slope of RDF at Kolmogorov length scale separations for higher St particles is not zero.

8:13AM M18.00002 A stochastic model of particle dispersion in turbulent reacting gaseous environments , GUANGYUAN SUN, DAVID LIGNELL, Brigham Young University, JOHN HEWSON, Sandia National Laboratory — We are performing fundamental studies of dispersive transport and time-temperature histories of Lagrangian particles in turbulent reacting flows. The particle-flow statistics including the full particle temperature PDF are of interest. A challenge in modeling particle motions is the accurate prediction of fine-scale aerosol-fluid interactions. A computationally affordable stochastic modeling approach, one-dimensional turbulence (ODT), is a proven method that captures the full range of length and time scales, and provides detailed statistics of fine-scale turbulent-particle mixing and transport. Limited results of particle transport in ODT have been reported in non-reacting flow. Here, we extend ODT to particle transport in reacting flow. The results of particle transport in three flow configurations are presented: channel flow, homogeneous isotropic turbulence, and jet flames. We investigate the functional dependence of the statistics of particle-flow interactions including (1) parametric study with varying temperatures, Reynolds numbers, and particle Stokes numbers; (2) particle temperature histories and PDFs; (3) time scale and the sensitivity of initial and boundary conditions. Flow statistics are compared to both experimental measurements and DNS data.

8:26AM M18.00003 Eulerian models for particle trajectory crossing in turbulent flows over a large range of Stokes numbers[1] , RODNEY O. FOX, Iowa State University and EM2C/Ecole Centrale Paris, AYMERIC VIE, FREDERIQUE LAURENT, Laboratoire EM2C - UPR CNRS 288, Ecole Centrale Paris, CHRISTOPHE CHALONS, Laboratoire Jacques-Louis Lions, Universite Pierre et Marie Curie - Paris 6, MARC MASSOT, Laboratoire EM2C - UPR CNRS 288, Ecole Centrale Paris — Numerous applications involve a disperse phase carried by a gaseous flow. To simulate such flows, one can resort to a number density function (NDF) governed a kinetic equation. Traditionally, Lagrangian Monte-Carlo methods are used to solve for the NDF, but are expensive as the number of numerical particles needed must be large to control statistical errors. Moreover, such methods are not well adapted to high-performance computing because of the intrinsic inhomogeneity of the NDF. To overcome these issues, Eulerian methods can be used to solve for the moments of the NDF resulting in an unclosed Eulerian system of hyperbolic conservation laws. To obtain closure, in this work a multivariate bi-Gaussian quadrature is used, which can account for particle trajectory crossing (PTC) over a large range of Stokes numbers. This closure uses up to four quadrature points in 2-D velocity phase space to capture large-scale PTC, and an anisotropic Gaussian distribution around each quadrature point to model small-scale PTC. Simulations of 2-D particle-laden isotropic turbulence at different Stokes numbers are employed to validate the Eulerian models against results from the Lagrangian approach. Good agreement is found for the number density fields over the entire range of Stokes numbers tested.

[1]Research carried out at the Center for Turbulence Research 2012 Summer Program

8:39AM M18.00004 Turbulence effects on particle dispersion in free-surface flow , SALVATORE LOVECCHIO, CRISTIAN MARCHIOLI, ALFREDO SOLDATI, University of Udine — We study the dispersion of light particles in a two-dimensional (2D) flow on a flat free-slip surface that bounds a three-dimensional (3D) volume in which the flow is turbulent. This simplified configuration mimics the motion of active/passive ocean surfactants (e.g. phytoplankton, floaters or drifters) when surface waves and ripples are absent. We perform direct numerical simulation of turbulence coupled with Lagrangian particle tracking, considering different values of the shear Reynolds number (Re=171 and 510) and of the Stokes number ($0.06 < \mathrm{St} < 1$ in viscous units). Simulations show that particles reach the free surface upon entrainment in upwelling motions, then quickly move toward downwelling regions where they are trapped for long residence times and advected by the mean flow. Surface flow is neither compressible nor incompressible but strongly influenced by the 3D flow underneath the surface. Results highlight the fractal nature of particle distribution at the surface, which appears to be driven by flow scales different from those of incompressible 2D/3D homogenous isotropic turbulence. In particular, we observe an asymptotic scaling of particle transport dynamics with the Lagrangian integral time scale of the fluid at the surface.

8:52AM M18.00005 A subgrid model for inertial particle clustering in large-eddy simulations of turbulence , BAIDURJA RAY, ANDREW D. BRAGG, LANCE R. COLLINS, Cornell University — Existing subgrid models for inertial particles in large-eddy simulations (LES) of turbulence do not correctly predict particle clustering. Synthetic turbulence models such as kinematic simulations (KS) have been shown to capture many features of fully developed turbulence, at low computational cost. The presence of small-scale flow structure (with a specified energy spectrum) makes KS an attractive choice for reconstructing the subgrid fluctuations seen by inertial particles in a LES. We apply such a model (referred to as KSSGM) to a filtered isotropic turbulence simulation with particles. Preliminary results show that the KSSGM is able to recover the RDF for moderately large Stokes numbers (for which clustering is still significant) and shows the correct qualitative trend in the RDF for smaller Stokes numbers (St). This suggests that the effect of subgrid scales on the high St particles is simpler than for particles having time-scales of the order of the Kolmogorov time-scale or less. Importantly, the KSSGM captures the opposing effect of the subgrid scales on clustering of particles with high and low St, which may stem from the fact that the KSSGM is able to describe the spatially correlated nature of the subgrid velocity field experienced by a particle pair.

9:05AM M18.00006 A comparison of theoretical models for the spatial clustering of inertial particles in homogeneous, isotropic turbulence , ANDREW BRAGG, LANCE COLLINS, Cornell University — In this talk we will consider two theoretical models for the clustering of inertial particles in turbulence, one by Chun *et.al.* (J. Fluid Mech. 536:219, 2005) and the other by Zaichik *et.al.* (Phys. Fluids. 19:113308, 2007). Although their predictions for the RDF (Radial Distribution Function) are similar in the regime $St \ll 1$ we will show that the two theories describe the physical origin of clustering in quite different ways. The Chun *et.al.* theory describes the origin of the clustering in terms of a local drift mechanism which arises because inertial particles sample more strain than rotation, and the Zaichik *et.al.* theory describes the origin of the clustering in terms of a drift mechanism which is influenced by both the local dynamics of the fluid velocity gradient tensor and also by the particle memory of its interaction with the fluid velocity field in its path history. We will discuss an artificial test case that demonstrates the physical mechanism described in the Chun *et.al.* theory does not completely describe the mechanism responsible for the clustering. Finally, we explain the discrepancies between the two theories and explain, despite those discrepancies, why their predictions in the regime $St \ll 1$ are so similar.

9:18AM M18.00007 A novel state-space based method for direct numerical simulation of particle-laden turbulent flows , REETESH RANJAN, CARLOS PANTANO, University of Illinois at Urbana-Champaign — We present a novel state-space-based numerical method for transport of the particle density function, which can be used to investigate particle-laden turbulent flows. Here, the problem can be stated purely in a deterministic Eulerian framework. The method is coupled to an incompressible three-dimensional flow solver. We consider a dilute suspension where the volume fraction and mass loading of the particles in the flow are low enough so that the approximation of one-way coupling remains valid. The particle transport equation is derived from the governing equation of the particle dynamics described in a Lagrangian frame, by treating position and velocity of the particle as state-space variables. Application and features of this method will be demonstrated by simulating a particle-laden decaying isotropic turbulent flow. It is well known that even in an isotropic turbulent flow, the distribution of particles is not uniform. For example, heavier-than-fluid particles tend to accumulate in regions of low vorticity and high strain rate. This lead to large regions in the flow where particles remain sparsely distributed. The new approach can capture the statistics of the particle in such sparsely distributed regions in an accurate manner compared to other numerical methods.

9:31AM M18.00008 Spatial effects of flow straining on inertial particles in turbulence , CHUNG-MIN LEE, California State University Long Beach, PRASAD PERLEKAR, FEDERICO TOSCHI, Eindhoven University of Technology, ARMANN GYLFASON, Reykjavik University — The effects of axisymmetric expansion on the movement of inertial particles are studied numerically. Turbulence with different strain rates is simulated with Direct Numerical Simulation and Rogallo's algorithm on a deforming domain, and particle movements are computed with the assumption of one-way coupling between the flow and particle fields. We are interested in the influence of the large scale geometric change on particle movements. We will present distribution results on inertial particles such as temporal correlations on particle locations, stagnation tendency, and encounters among particles with different Stokes numbers.

9:44AM M18.00009 Particle tracking in LES flow fields: Lagrangian conditional statistics of filtering error , MARIO TESONE, Univesrity of Pisa, MARIA VITTORIA SALVETTI, University of Pisa, CRISTIAN MARCHIOLI, University of Udine, SERGIO CHIBBARO, Pierre et Marie Curie University - Paris 6, ALFREDO SOLDATI, University of Udine — The Lagrangian PDFs of the fluid velocity filtering error associated to Lagrangian particle tracking in filtered DNS flow fields are examined. To this aim, we perform a-priori tests in which the error purely due to filtering is singled out removing error accumulation effects, which would lead to progressive divergence between DNS and filtered DNS trajectories. PDFs are then obtained for the reference case of turbulent channel flow, conditioning the initial particle distribution within regions where either a sweep or an ejection is taking place. Preliminary results confirm the stochastic and non-Gaussian nature of filtering error in non-homogeneous flows. Compared to Eulerian PDFs, however, Lagrangian conditional PDFs exhibit differences which may offer useful insights for physical modelling. Specifically, for short times upon particle release, PDFs indicate a strong subgrid anisotropic effect of sweeps and ejections along the wall-normal direction. This feature underlines the link between turbulent coherent structures and strain, suggesting the possibility to model coherent structures with a direct link to velocity gradients. Asymptotically, the Lagrangian conditional PDFs recover the Eulerian behavior showing Stokes number effects limited to the PDF tails.

9:57AM M18.00010 A numerical investigation of cluster fall velocity in vertical particle-laden turbulent pipe flow , JESSE CAPECELATRO, OLIVIER DESJARDINS, Cornell University, NATIONAL RENEWABLE ENERGY LAB COLLABORATION — Particle clusters are known to play a key role in the multiphase dynamics as well as secondary processes such as heat transfer and catalytic conversion within vertical pipe flows. For example, vertical risers in circulating fluidized bed reactors consists of a dilute suspension of particles that ascend in the core of the flow, then condense into clusters and descend at the walls. In this work, an Euler-Lagrange strategy is used to study particle cluster dynamics in turbulent risers for a range of Archimedes numbers and density ratios. The simulations are conducted in the framework of NGA, a high-order fully conservative code tailored for turbulent flows. The particles are solved in a Lagrangian framework and the two phases are coupled using a two-step filtering process to ensure conservation, as well as convergence during mesh refinement. Normal and tangential collisions are computed via a soft-sphere model. A conservative immersed boundary method is used to represent the 3D cylindrical geometry on a Cartesian mesh. Simulation results are compared with experimental correlations in terms of cluster fall velocity and size. The role of the carrier fluid on the cluster behavior is also studied.

Tuesday, November 20, 2012 8:00AM - 10:10AM –

Session M19 Non-Newtonian Flows III 28E - Chair: Gareth McKinley, Massachusetts Institute of Technology

8:00AM M19.00001 Flow-induced structures in polymer-colloid mixtures with attractive interactions , PATRICK UNDERHILL, RANGARAJAN RADHAKRISHNAN, Rensselaer Polytechnic Institute — In some polymeric or surfactant systems, structures can form in flow that would not normally form at equilibrium. This can occur for hydrophobic poylelectrolytes, associative polymers, wormlike micelles, and mixtures of polymers and nanoparticles. These systems are particularly challenging to model because of the separation of time and length scales between the polymer radius of gyration and conformational rearrangements and the scale of the attractive and repulsive interactions. We have developed a new coarse grained model of polymer solutions that show flow-induced structures and have performed Brownian Dynamics simulations. These simulations provide insight into the mechanisms for forming flow-induced structures and the similarities between different systems. We will discuss how the effective solvent quality and balance of attractive and repulsive interactions alter the flow transitions and whether the structures are reversible or irreversible. We have also analyzed the dependence on the flow type. In particular, we have looked at oscillatory shear and elongational flows.

8:13AM M19.00002 On singularity of the UCM and Oldroyd-B models in viscoelastic fluids: resolving the high-Weissenberg number problem[1] , RHO SHIN MYONG, Gyeongsang National University — Most of methods based on the UCM and Oldroyd-B models in viscoelastic fluids are found to break down at a frustratingly low value of the Weissenberg number around We=1. The rigorous explanation for this mysterious break-down has remained elusive until recently. In this work, the nature of mathematical singularity of these classical models is first elucidated by considering shear, compression, and extension flows. Then a regularization method based on the Rayleigh-Onsager quadratic dissipation function is proposed in order to resolve the high-Weissenberg number problem. In particular, the exact reason why the extensional flow suffers the break-down in high-Weissenberg number cases is explained. In addition, the relationship of the regularized model to other constitutive models such as the Giesekus and the Phan-Thien-Tanner equations is illustrated.

[1]This work was supported by the Degree and Research Center for Aerospace Green Technology (DRC) funded by the Korea Research Council of Fundamental Science & Technology (KRCF).

8:26AM M19.00003 Flow of a viscous nematic fluid around a microsphere , MANUEL GOMEZ-GONZALEZ, JUAN C. DEL ALAMO, University of California, San Diego — We analyze the creeping flow elicited by a rigid spherical particle in a viscous nematic fluid. The drag force acting on the particle and its far flow velocity field are obtained in closed analytical form. We identify two anisotropy mechanisms arising from the constitutive equations, namely, the anisotropic momentum diffusivity and the resistance to bending of the fluid. Their influence on the flow is studied separately by defining two "pure" pseudo-isotropic conditions in which only one of these mechanisms is present. The accuracy of existing Particle Tracking Microrheology methods is analyzed in these fluids, with especial attention to an approach that provides effective directional viscosity coefficients by applying Stokes's drag separately along different directions. Finally, we analyze the effect of anisotropy on the flow structure, finding that directional momentum diffusivity produces asymmetric streamline patterns with enhanced streamline deflection in the directions of lower diffusivity and viceversa. This asymmetry can either be increased or decreased by the bending resistance of the fluid, depending on the sign of the bending stresses. In some cases, their combined effect is found to produce streamline patterns that converge towards the sphere.

8:39AM M19.00004 Experiments on Rearrangements and Forces in 2D Emulsion Hopper Flow , XIA HONG, KENNETH DESMOND, DANDAN CHEN, ERIC WEEKS, Physics Department, Emory University — We conduct experiments with a quasi-two-dimensional binary emulsion flowing through a hopper. Our samples are oil-in-water emulsions confined between two close-spaced parallel plates, so that the droplets are deformed into pancake shapes. In this system, there is only viscous friction and no static friction between droplets. The hopper flow induces a high rate of rearrangement events allowing us to understand how stresses and forces change during the process. By imaging the droplets during flow, we observed T1 events, which are topological changes when droplets exchange neighbors. Simultaneously, we measure forces between the droplets using a technique we have developed. We study the evolution of forces between droplets during rearrangements, finding that rearrangements decrease the inter-droplet forces of nearby droplets.

8:52AM M19.00005 Response of a two-dimensional liquid foam to air injection: swelling rate, fingering and fracture , BENJAMIN DOLLET, CNRS, IMEN BEN SALEM, ISABELLE CANTAT, Université Rennes 1 — The response of a two-dimensional liquid foam to a localised air injection is investigated experimentally and theoretically. The experiments show a rich phenomenology, with two essentially distinct behaviours, depending on the injection conditions. At low flux, the injected air forms a central bubble that grows inside the foam and induces plastic rearrangements, without film rupture. This "pure swelling" regime is reminiscent of ductile fracture. In this regime, the central bubble shows fingering patterns beyond a certain velocity. The dependence between the swelling rate, the injection overpressure and the other control parameters: cell gap, bubble size and foam area, is captured by a simple balance between the pressure drop and bubble/wall friction within a radial assumption. Fingering is successfully modelled by the linear stability analysis of an azimuthal perturbation of the radial model; yield stress becomes an important parameter to determine the finger width. At high injection rate, films are broken and narrow cracks form rapidly through the foam, which reminds brittle fracture. Criteria on the transition between ductile and brittle behaviours are investigated, both at the local and global scales.

9:05AM M19.00006 Sedimentation of solid spheres and Newtonian drops in a viscoplastic medium , OLGA LAVRENTEVA, YULIA HOLENBERG, Chemical Engineering Dept., Technion, URI SHAVIT, Envinromental, Water and Agricultural Engineering Dept., Technion, AVINOAM NIR, Chemical Engineering Dept., Technion — The slow sedimentation of smooth and rough solid spheres and Newtonian drops in viscoplastic fluid (low concentrated, of 0.07% w/w, aqueous gel of Carbopol 940) is studied experimentally. It was found that when the drops settle in proximity to a vertical solid wall their settling speed is augmented and drops drift slowly toward the wall. This is contrary to what is known with Newtonian or viscoelastic domains. The shape of yielded regions around the particles and the flow field within these regions were determined making use of PTV and PIV methods. It is observed that yielded region around slowly moving rough sphere is almost fore-and-aft symmetric and resembles that obtained via numerical modeling. In contrast to this, yielded regions around drops and smooth particles are not symmetric. A striking similarity between flow fields generated by the motion of the later two types of inclusions was observed. This phenomenon as well as the increase of settling speed in the vicinity of walls can be attributed to the dynamic formation of a thin clear layer providing effective slip at the interface of smooth particles or at the wall. The existence of the wall slip was demonstrated also in our studies of Carbopol gel rheology.

9:18AM M19.00007 Dynamics of Non-Newtonian Liquid Droplet Collision[1] , XIAODONG CHEN, VIGOR YANG, Georgia Institute of Technology — Collision of Newtonian liquid droplets has been extensively investigated both experimentally and numerically for decades. Limited information, however, is available about non-Newtonian droplet collision dynamics. In the present work, high-fidelity numerical simulations were performed to study the situation associated with shear-thinning non-Newtonian liquids. The formulation is based on a complete set of conservation equations for the liquid and the surrounding gas phases. An improved volume-of-fluid (VOF) method, combined with an innovative topology-oriented adaptive mesh refinement (TOAMR) technique, was developed and implemented to track the interfacial dynamics. The complex evolution of the droplet surface over a broad range of length scales was treated accurately and efficiently. In particular, the thin gas film between two approaching droplets and subsequent breakup of liquid threads were well-resolved. Various types of droplet collision were obtained, including coalescence, bouncing, and reflexive and stretching separations. A regime diagram was developed and compared with that for Newtonian liquids. Fundamental mechanisms and key parameters that dictate droplet behaviors were identified. In addition, collision-induced atomization was addressed.

[1]This work was sponsored by the U.S. Army Research Office under the Multi-University Research Initiative under contract No. W911NF-08-1-0124. The support and encouragement provided by Dr. Ralph Anthenien are gratefully acknowledged.

9:31AM M19.00008 Lateral migration of a viscoelastic drop in a shear flow near a wall[1] , SWARNAJAY MUKHERJEE, University of Delaware, KAUSIK SARKAR, George Washington University — Deformation, orientation and lateral migration of a viscoelastic drop suspended in a wall-bounded shear flow of Newtonian fluid is numerically investigated using a front-tracking finite-difference method. The viscoelasticity is modeled using the modified FENE-CR constitutive equation. The initial position is found to not affect the quasi-steady migration dynamics of a viscoelastic drop just like that of a Newtonian drop. As viscoelasticity is increased, lateral migration is enhanced initially, then a saturation is reached and finally for very high viscoelasticity lateral migration decreases. This non-monotonicity is due to the presence of two opposite factors: the interfacial stresslet term and the non-Newtonian normal stresses inside the drop phase. Viscoelasticity increases the orientation angle which hinders migration by decreasing the interfacial stresslet term. Deformation is non-monotonic affecting the stresslet term and adding to the non-monotonicity of migration. Finally, the viscoelastic normal stresses inside promote migration in contrast to the case of a Newtonian drop migrating in a viscoelastic matrix, where the viscoelastic stresses outside hinders migration. The viscoelastic effect is enhanced at lower Ca and higher viscosity ratios.

[1]Partially supported by NSF.

9:44AM M19.00009 The effects of low Reynolds number viscoelasticity on linked sphere swimmers , MARK CURTIS, EAMONN GAFFNEY, None — A simple model for a swimmer immersed in a zero Reynolds number environment consisting of three linked spheres attached by extensible rods contracting out of phase to break reciprocal motion is analysed. By prescribing the forces acting on the three spheres due to the rods, asymptotic methods are used to derive analytic expressions for the net displacement of the swimmer in both a Newtonian Stokes fluid and a zero Reynolds number viscoelastic fluid. The model indicates that the swimmer, during one beat cycle, can actually move a greater distance when immersed in the viscoelastic fluid compared to the Newtonian fluid.

9:57AM M19.00010 Droplet Size Distributions in Atomization of Dilute Viscoelastic Solutions , BAVAND KESHAVARZ, GARETH MCKINLEY, M.I.T. Mechanical Engineering Department, ERIC HOUZE, JOHN MOORE, MICHAEL POTTIGER, PATRICIA COTTS, DuPont, M.I.T. COLLABORATION[1], DUPONT COLLABORATION[2] — The droplet size probability distribution functions (PDF) for atomization/fragmentation processes in Newtonian fluids are now generally accepted to be close to Gamma distributions. Despite the great practical importance, little is known about the nature of corresponding distributions for viscoelastic liquids, e.g. polymeric solutions such as pesticide sprays and paints. We present data from air-assisted atomization experiments for model viscoelastic solutions composed of very dilute solutions of polyethylene oxide. Although the addition of small amounts of high molecular weight polymer keeps the fluid shear viscosity and surface tension close to the solvent values, the size distributions are skewed towards higher values of the Sauter mean diameter. We show that the PDF curves for these weakly-elastic fluids are well described by Gamma distributions, but the exponent *n* is systematically decreased by fluid elasticity. Flow visualization images show that this behavior arises from the non-linear dynamics close to the break-up point which are dominated by an elasto-capillary force balance within the thinning ligaments and the magnitude of the extensional viscosity in the viscoelastic fluid.

[1]Mechanical Engineering Department, Cambridge, MA.
[2]Experimental Site, Delaware

Tuesday, November 20, 2012 8:00AM - 10:10AM –

Session M20 Turbulent Boundary Layers X: Drag Reduction 30A - Chair: Jim Brasseur, Pennsylvania State University

8:00AM M20.00001 Effects of Superhydrophobic Surface on Laminar and Turbulent Flows[1] , HYUNWOOK PARK, JOHN KIM, University of California, Los Angeles — The recent development of superhydrophobic surfaces (SHSs) has attracted much attention as it leads to a possibility of achieving substantial skin-friction drag reduction at high Reynolds number (Re) turbulent flows. A SHS, consisting of a hydrophobic surface combined with micro- or nano-scaled topological features, can yield an effective slip length on the order of several hundred microns. In this presentation, the effects of SHSs on both laminar and turbulent channel flows are investigated numerically. A SHS is modeled through no-slip boundary condition on top of micro-scaled features and stress-free boundary condition on gas-liquid interfaces. In laminar flows, the effective slip length depends on the geometry only independent of Re, consistent with the analysis of Lauga and Stone (2003)[2], while in turbulent flows it depends on Re, thus indicating its dependence on flow conditions near the surface. The resulting drag reduction is much larger in turbulent flows and near-wall turbulence structures were significantly modified. We conclude that this indirect effect plays a more significant role in reducing drag in turbulent flows than the direct effect of the shear-free condition that led to drag reduction in laminar flows.

[1]supported by ONR grant, N000141110503.
[2]E. Lauga and H.A. Stone, JFM 489 (2003) 55-77

8:13AM M20.00002 Direct measurement of turbulent skin-friction reduction on superhydrophobic surfaces[1] , HYUNGMIN PARK, GUANGYI SUN, CHANG-JIN "CJ" KIM, University of California, Los Angeles — Recent advances in superhydrophobic (SHPo) surfaces have spurred a great interest in fluid mechanics because their large slip may result in a significant reduction of skin friction in turbulent flows. However, experimental confirmation of the reduction has been sporadic (only internal flows) and equivocal because most times the surface slip was small and the drag measurement indirect. Here we present a direct measurement of the drag on large-slip surfaces in a turbulent boundary-layer flow. The silicon-micromachined sample has a SHPo (microgrates) next to a reference (smooth) surface, each suspended by identical micro flexure beams. Monolithically fabricated in a batch process and sharing all the variations, the two surfaces shift differently only by the difference in the drag. The drag reduction was measured optically (directly) in a turbulent boundary layer in a water tunnel experiment at a moderate Reynolds number ($Re_\tau \sim 250$) over a gas fraction (fraction of the shear-free surface area) of $30\% - 90\%$. Unlike other reports, the drag reduction clearly increased with the gas fraction. More than 50% skin-friction reduction was achieved with 90% gas fraction. During the flow tests, the SHPo surfaces were visually confirmed to contain the air without any loss.

[1]Supported by the Office of Naval Research (ONR) Program (N000141110503).

8:26AM M20.00003 Direct numerical simulation of turbulent flows over superhydrophobic surfaces with gas pockets using linearized boundary conditions[1] , JONGMIN SEO, SANJEEB BOSE, RICARDO GARCIA-MAYORAL, ALI MANI, Stanford University — Superhydrophobic surfaces are shown to be effective for surface drag reduction under laminar regime by both experiments and simulations (see for example, Ou and Rothstein, Phys. Fluids 17:103606, 2005). However, such drag reduction for fully developed turbulent flow maintaining the Cassie-Baxter state remains an open problem due to high shear rates and flow unsteadiness of turbulent boundary layer. Our work aims to develop an understanding of mechanisms leading to interface breaking and loss of gas pockets due to interactions with turbulent boundary layers. We take advantage of direct numerical simulation of turbulence with slip and no-slip patterned boundary conditions mimicking the superhydrophobic surface. In addition, we capture the dynamics of gas-water interface, by deriving a proper linearized boundary condition taking into account the surface tension of the interface and kinematic matching of interface deformation and normal velocity conditions on the wall. We will show results from our simulations predicting the dynamical behavior of gas pocket interfaces over a wide range of dimensionless surface tensions.

[1]Supported by the Office of Naval Research and the Kwanjeong Educational Scholarship Foundation.

8:39AM M20.00004 Pressure Drop Measurements for Turbulent Channel Flow over Superhydrophobic Surfaces with Superimposed Riblets , RICHARD PERKINS, JOSEPH PRINCE, JULIE VANDERHOFF, DANIEL MAYNES, Brigham Young University — We consider the combined drag reducing mechanisms of riblets and superhydrophobicity. Pressure drop measurements were acquired for turbulent channel flow over riblet surfaces, superhydrophobic surfaces, and surfaces with both drag reducing mechanisms. The riblets were nominally 80 μm tall, 16 μm wide, and spaced with a period of 160 μm. The superhydrophobic structuring was composed of alternating microribs (15 μm tall and 8 μm wide) and cavities (32 μm wide), aligned parallel to the flow. The channel consisted of a control section and a test section comprised of smooth and patterned wafers, respectively. In all cases, the test section walls were structured on top and bottom while the side walls were left smooth. The channel had a hydraulic diameter of 7.3 mm and an aspect ratio of 10:1. Seven pressure ports were precision machined into the walls of both the control and test sections. The pressure drop measurements were acquired simultaneously over both sections to eliminate uncertainty associated with the flow rate. The drag reduction for all test sections was then computed directly and data were obtained over a Reynolds number range of 11000 to 15000.

8:52AM M20.00005 Dual-plane stereoscopic PIV measurements of turbulence statistics in drag-reducing pipe flows with surfactant additive , YOSHITSUGU NAKA, SHUHEI NUKA, MASAYASU SHIMURA, NAOYA FUKUSHIMA, MAMORU TANAHASHI, TOSHIO MIYAUCHI, Tokyo Institute of Technology — Drag-reducing turbulent pipe flows with surfactant additive are investigated by dual-plane stereoscopic PIV (DPSPIV). The DPSPIV system gives two parallel planes of three component velocity with a good spatial resolution. Measurements are undertaken in a fully developed pipe flow for 21 different conditions, i.e., the combinations of 3 concentrations of the surfactant additive and 7 Reynolds numbers. Two stereo PIV planes are settled at $50D$ downstream from the pipe inlet and are arranged perpendicular to the mean flow with a small spacing in the streamwise direction. Within the present Re range, the drag-reduction ratio takes its maximum value 63% at Re_D=56 000. Noticeable differences between drag-reducing and water flows are observed on their statistics. The peaks of the rms values of the velocity fluctuations are shifted away from the wall since the thicknesses of the viscous sub-layer and the buffer region increase compared to the water flow case. In the shear stress profile, the Reynolds shear stress is much reduced in the near wall region, and the viscous shear stress contributes more. Velocity gradient characteristics such as profiles of vorticity components and the balance of the transport equation of the turbulent kinetic energy are also described.

9:05AM M20.00006 New approaches to turbulent skin-friction drag reduction in wall flows based on the mechanism of drag reduction of polymer additives , RAYHANEH AKHAVAN, DONG-HYUN LEE, University of Michigan, Ann Arbor, MI, 48109-2125 — Polymer additives provide one of the most effective means of skin-friction DR in wall flows. Recent insights into the DR mechanism of polymer additives reveals that the cornerstone of high DR with polymer additives is redirection of a small fraction of TKE from energy-containing eddies into a path other than the normal turbulent energy cascade. This redirection of energy initiates a self-amplifying sequence of events within the turbulence dynamics, through which the turbulence loses its three-dimensional structure and turbulence production is suppressed. This mechanism suggests that the same dramatic DRs observed with polymer additives can be reproduced through *any* mechanism which redirects a small fraction of TKE from energy-containing eddies into a path other than the normal turbulent energy cascade. This hypothesis has been tested using DNS in turbulent channel flow, where the molecular viscosity in a small band of wavenumbers, corresponding to $0.01 < k/k_d < 0.1$, was increased from ν to 4ν. Drag reductions of up to 54%, comparable to that observed with polymers at MDR, were observed, and many of the flow features observed in DR with polymer additives were reproduced. These results open up new possibilities for devising novel DR strategies for wall flows.

9:18AM M20.00007 The spectral link for frictional drag in non-uniform turbulent soap-film flows[1] , CHIEN-CHIA LIU, Okinawa Institute of Science and Technology, RORY CERBUS, WALTER GOLDBURG, University of Pittsburgh, GUSTAVO GIOIA, PINAKI CHAKRABORTY, Okinawa Institute of Science and Technology — The spectral link provides a novel relation between the frictional drag (f) and the exponent α of the turbulent energy spectrum ($E(k) \sim k^{-\alpha}$): $f \sim \mathrm{Re}^{(1-\alpha)/(1+\alpha)}$, where Re is the Reynolds number. The spectral link has been verified experimentally in uniform turbulent soap-film flows where the same α, which is either 3 (the enstrophy cascade) or 5/3 (the inverse energy cascade), prevails across the width of the flow. We perform experiments on non-uniform turbulent soap-film flows where α switches between 3 and 5/3 across the width of the flow. Our measurements of f vs. Re are in excellent accord with the localized version of the spectral link.

[1]This research was partially supported by funds from the Okinawa Institute of Science and Technology

9:31AM M20.00008 Drag reduction in Turbulent Channel Flow with Longitudinal Arrays of Slip/no-slip Stripes on the Walls , AMIRREZA RASTEGARI, RAYHANEH AKHAVAN, University of Michigan, Dept. of Mechanical Engineering, Ann Arbor, MI 48109-2125 — Drag reduction in channels covered with longitudinal arrays of slip/no-slip stripes on the walls has been investigated using DNS with the lattice Boltzmann method. Computations were performed in channels of size $5h \times 2.5h \times 2h$ at a $Re_b = 3600$ ($Re_{\tau 0} \approx 230$) with stripes of size $0.02 \leq g/h = w/h \leq 0.56$ corresponding to $4 \leq g^{+0} = w^{+0} \leq 128$ where $g = w$ denotes the widths of the slip/no-slip stripes and h is the channel half-width. Unlike in laminar flow, where the magnitude of DR is controlled by geometrical parameters g/h and w/h, in turbulent flow the magnitude of DR is found to scale with $g^{+0} = w^{+0}$, independent of Reynolds number. DRs of $5\%, 11\%, 18\%, 23\%, 38\%, 47\%$, and slip velocities of $U_s/U_b = 0.06, 0.10, 0.15, 0.23, 0.32, 0.37$ were observed for $g^{+0} = w^{+0} = 4, 8, 16, 32, 64, 128$, respectively. Analysis of the mechanism of DR reveals that in the LDR regime ($DR < 25\%$, $g^{+0} \leq 32$, $U_s/U_b < 0.25$), DR is due to a combination of wall-slip and change in the anisotropy structure of turbulence near the wall, while in the HDR regime ($DR > 30\%$, $g^{+0} \geq 64$, $U_s/U_b > 0.3$), DR is primarily due to cessation of turbulence production over the slip stripes due to the large slip velocities over these regions.

9:44AM M20.00009 Experimental investigation for drag reduction effect by traveling wave-like wall deformation in turbulent channel flow , HIROYA MAMORI, Keio University, YUHO ISHIWATA, KAORU IWAMOTO[1], AKIRA MURATA, Tokyo University of Agriculture and Technology — A traveling wave-like control for wall-turbulence is known to have a potential to relaminarize a turbulent flow to a laminar flow, which has been confirmed by means of a direct numerical simulation. Especially, a wall deformation is one of the methods to realize this traveling wave-like control. The objective of the present study is an investigation for the drag reduction effect due to the traveling wave-like wall deformation experimentally. An elastic rubber is used for a vibrating plate and an oscillator is composed by an amplified piezoelectric actuator. The parameter of the traveling wave is preliminary determined by the direct numerical simulation, which induces the relaminarization at low-to-moderate Reynolds numbers. The drag coefficient is measured in the range of $2000 < \mathrm{Re_b} < 10000$, where $\mathrm{Re_b}$ is the bulk Reynolds number. The traveling wave-like wall deformation is found to decrease the drag below the uncontrolled turbulent flow level. The dependency of the drag coefficient for the frequency of the vibrating of the actuator is also investigated. The flow field is visualized by a Particle Image Velocitmetry measurement, and the drag reduction mechanism is discussed by using the turbulent statistics of the controlled flow.

[1]corresponding author

9:57AM M20.00010 Equilibrium turbulent boundary layers with wall suction/blowing and pressure gradients , SAURABH PATWARDHAN, O.N. RAMESH, Indian Institute of Science — Conditions for the equilibrium conditions in turbulent boundary layers with suction or blowing across a no slip wall and pressure gradients are derived from the governing equations. It is also shown that under these conditions the governing equations show self similarity in the conventional inner co-ordinates as well as "laminar-like" co-ordinates. The only turbulent boundary layer in "perfect equilibrium" known as sink flow turbulent boundary layer forms a subset of this more general equilibrium concept. Direct numerical simulations were carried out to investigate this hypothesis for the case of favourable pressure gradient with small blowing at the wall. Reynolds number invariance and complete self similarity of mean velocity profile and second order turbulence statistics is observed along the flow direction similar to the sink flow boundary layer. A comparison between the case with wall blowing and imposed favourable pressure gradient and the sink flow case for same value of pressure gradient parameter reveals a shift in log law in mean velocity profile and increase in peak turbulence intensities.

Tuesday, November 20, 2012 8:00AM - 10:10AM –

Session M21 Turbulence Shear Layer 30B - Chair: Joseph Katz, Johns Hopkins University

8:00AM M21.00001 ABSTRACT WITHDRAWN –

8:13AM M21.00002 Momentum balance in wall jets , T. GUNNAR JOHANSSON, Applied Mechanics Dept., Chalmers Univ of Technology, Göteborg, Sweden, FARAZ MEHDI, Dept. of Mechanical Engineering, Univ. of New Hampshire, Durham, NH, JONATHAN W. NAUGHTON, Dept. of Mechanical Engineering, Univ. of Wyoming, Laramie, WY — A plane wall jet experiment has been done to study its momentum balance. Two component laser Doppler anemometry was used to simultaneously measure the axial and wall-normal velocity components in 6 axial positions (x/H= 25, 50, 75, 100, 125 and 150) spanning from the wall all the way well into the ambient stagnant area. In this way not only the mean velocity components and Reynolds normal and shear stresses but also all their spatial derivatives were determined. In addition the wall shear stress was measured in all six axial positions using oil film interferometry. From these data all terms in the x-momentum equation, except the pressure term, could be evaluated. Later also the pressure was measured in the same profiles, and thereby also the pressure term was included in the balance. Contrary to common belief it was found that the pressure was not constant in the wall jet. The complete momentum balance is discussed and used to evaluate the roles played by the different contributing terms in different regions of the flow field in an effort to improve on our understanding of the mechanics of wall jets.

8:26AM M21.00003 Statistics of the local turbulent/nonturbulent interface thickness in jets and boundary layers , RODRIGO TAVEIRA, CARLOS DA SILVA, IST - Technical University of Lisbon, GUILLEM BORRELL, JAVIER JIMENEZ, School of Aeronautics, Universidad Politecnica de Madrid — Direct numerical simulations (DNS) of turbulent planar jets and boundary layers are used to assess and compare the differences/similarities in the turbulent/nonturbulent (T/NT) interface separating the turbulent from the irrotational flow regions in these flows. Specifically, we focus on the local thickness and local vorticity magnitude across the T/NT interface. Probability density functions of the local T/NT interface thickness in jets show the existence of a plateau between 3 to 10 Kolmogorov micro-scales wide, while the mean thickness is of the order of the Taylor micro-scale, as observed in recent studies. Between 60 to 70% of the local thickness values are among this range. On the other hand only 5% of the local vorticity magnitudes are within the range designated by intense vorticity structures ("worms") and around 60% of the local vorticity magnitudes after the T/NT interface is in the range of background to weak turbulence. The goal of the work is to characterise what type of structures (vortices/sheets) and in what fraction do they make up the T/NT interface. A similar analysis for a boundary layer is under way.

8:39AM M21.00004 Eulerian and lagrangian statistics across the turbulent/non-turbulent interface in turbulent jets , JOSÉ ALEXANDRE DIOGO, RODRIGO TAVEIRA, CARLOS DA SILVA, IST - Technical University of Lisbon — Direct numerical simulations (DNS) of turbulent planar jets are used to obtain Eulerian and Lagrangian statistics of the flow across the turbulent/nonturbulent (T/NT) interface, separating the turbulent and the irrotational flow regions. The Eulerian statistics used here consist in conditional statistics made in relation to the distance from the T/NT interface that eliminate the large scale intermittence affecting classical eulerian statistics near the jet edge. For the Lagrangian statistics we use tracers and study their statistics during the entrainment. We focus on the enstrophy characteristics and on the mechanism of enstrophy generation and associated time and length scales as well as on the relative importance between engulfing and nibbling mechanisms.

8:52AM M21.00005 Small scale entrainment characteristics in variable density turbulent jets , PEDRO COSTA, CARLOS DA SILVA, IST - Technical University of Lisbon — Direct numerical simulations (DNS) of turbulent planar jets with different densities in the irrotational and in the turbulent region are used to assess the small scale characteristics associated with the entrainment in turbulent jets. The mean thickness of the enstrophy jump at the turbulent/nonturbulent (T/NT) interface separating the turbulent and the irrotational flow regions is slightly bigger in variable density than in incompressible jets, which may be attributed to the more elongated shape of the coherent vortices at that location. Moreover, the characteristic velocity jump is also bigger which can be explained by a momentum balance made in a control volume near the T/NT interface, as done in recent works for the incompressible case. Finally, variable density jets also display smaller (negative) mean entrainment conditional velocity profiles compared to the incompressible case.

9:05AM M21.00006 Extreme events in a vortex gas simulation of a turbulent half-jet[1] , SAIKISHAN SURYANARAYANAN, JNCASR, Bangalore, GOKUL PATHIKONDA[2], University of Illinois at Urbana-Champaign , RODDAM NARASIMHA, JNCASR, Bangalore — Extensive simulations [arXiv:1008.2876v1 [physics.flu-dyn], BAPS.2010.DFD.LE.4] have shown that the temporally evolving vortex gas mixing layer has 3 regimes, including one which has a universal spreading rate. The present study explores the development of spatially evolving mixing layers, using a vortex gas model based on Basu et al (1995 Appl. Math. Modelling). The effects of the velocity ratio (r) are analyzed via the most extensive simulations of this kind till date, involving up to 10000 vortices and averaging over up to 1000 convective times. While the temporal limit is approached as r approaches unity, striking features such as extreme events involving coherent structures, bending, deviation of the convection velocity from mean velocity, spatial feedback and greater sensitivity to downstream and free stream boundary conditions are observed in the half-jet (r = 0) limit. A detailed statistical analysis reveals possible causes for the large scatter across experiments, as opposed to the commonly adopted explanation of asymptotic dependence on initial conditions.

[1]Supported in part by contract no. Intel/RN/4288

[2]Summer Research Intern at JNCASR via RISE, Indo-US Science and Technology Forum.

9:18AM M21.00007 A revisit of the low-frequency flapping mechanism in an open cavity shear layer flow[1] , XIAOFENG LIU, JOSEPH KATZ, Johns Hopkins University — Causes for low-frequency flapping in an open cavity shear layer flow are revisited using correlations among experimentally obtained time-resolved velocity and pressure distributions. Interactions of the shear layer with the downstream corner cause substantial local variations in pressure and velocity, but correlations show little evidence that these phenomena affect the flow around the upstream corner. Instead, it seems that the flapping occurs due to changes in the streamwise pressure gradients ($\partial p'/\partial x$) associated with vertical motion of the shear layer in the vicinity of the upstream corner. When the highly correlated initial part of the shear layer and the boundary layer upstream of the corner are thin, $\partial p'/\partial x$ increases. The elevated adverse pressure gradients increase the thickness of the boundary layer, causing a downstream propagating increase in shear layer elevation. As the shear layer rises, $\partial p'/\partial x$ around the upstream corner decreases, causing thinning of the boundary layer. This process persists at $St{=}fL/U_\infty$ of 0.052, an order of magnitude lower than that associated with transport of shear layer vortices. Oscillations in the vertical velocity along the upstream cavity wall are weakly correlated with the flapping except for a small separation bubble near the corner.

[1]Sponsored by ONR and NSF.

9:31AM M21.00008 Linear analysis and temporal DNS of compressible mixing layers , MONA KARIMI, SHARATH GIRIMAJI, Texas A&M University — We perform linear analysis and temporal direct numerical simulations (DNS) of high-speed compressible mixing layers. The DNS solver is based on Gas Kinetic Method (GKM) and has been validated in a variety of high-speed shear flows and against rapid distortion theory results. The objective is to examine the effect of compressibility on Kelvin-Helmholtz instability. We perform modal and statistical analysis. The difference in the behavior of two-dimensional and oblique modes is investigated. The action of pressure on 2D and oblique modes is differentiated. The stabilizing influence of compressibility on mixing layer growth (quantified by the so-called Langley Curve) in investigated from fundamental principles.

9:44AM M21.00009 Time-resolved PIV measurements of flow around and through a permeable rectangular prism , G.L. BLOIS, J.M. BARROS, J.L. BEST, K.T. CHRISTENSEN, University of Illinois at Urbana-Champaign — The unsteady turbulent wake dynamics generated by flow around and through a permeable rectangular prism is experimentally investigated using a refractive-index-matching (RIM) approach. A 7-mm diameter cylindrical flow passage was drilled through the center of an acrylic rectangular prism (25.5-mm thick; 51.0-mm long) along its streamwise axis. This permeable prism was then immersed in an aqueous solution ($\sim 63\%$ by weight) of sodium iodide (NaI) within a recirculating flow loop wherein the refractive index of the NaI was accurately matched to that of the acrylic prism via control of the NaI concentration and solution temperature. This RIM approach enabled simultaneous optical interrogation of the flow both around and within the permeable prism with time-resolved particle-image velocimetry at $\mathrm{Re} > 10^4$. The interaction between the flow exiting the passage and the vortices shed from the model dramatically modifies the dynamics of the wake compared to that of a solid prism of identical dimensions. While the flow through the passage was found to be relatively steady, it generated a pulsating jet that penetrated into the wake, yielding strong internal–external flow interactions.

9:57AM M21.00010 Measurement of the flow past a cactus-inspired cylinder , GHANEM F. OWEIS, ADNAN M. EL-MAKDAH, Mechanical Engineering, American University of Beirut, Beirut, Lebanon — Desert cacti are tall cylindrical plants characterized by longitudinal u- or v-shaped grooves that run parallel to the plant axis, covering its surface area. We study the wake flow modifications resulting from the introduction of cactus-inspired surface grooves to a circular cylinder. Particle image velocimetry PIV is implemented in a wind tunnel to visualize and quantify the wake flow from a cactus cylinder in cross wind and an equivalent circular cylinder at Re O(1E5). The cactus wake exhibits superior behavior over its circular counterpart as seen from the mean and turbulent velocity profiles. The surface flow within the grooves is also probed to elucidate the origins of the wake alterations. Lastly, we use simple statistical analysis based only on the wake velocity fields, under the assumption of periodicity of the shedding, to recover the time varying flow from the randomly acquired PIV snapshots.

Tuesday, November 20, 2012 8:00AM - 9:44AM –

Session M22 Turbulence Mixing IV 30C - Chair: Diego Donzis, Texas A&M University

8:00AM M22.00001 Experimentally Informed Turbulent Diffusivity Modeling for an Angled Jet in Cross-Flow , JULIA LING, FILIPPO COLETTI, SAYURI YAPA, GIANLUCA IACCARINO, JOHN EATON, Stanford University — A key source of error in many turbulent heat transfer simulations is the modeling of the turbulent heat flux. This heat flux is often approximated using the gradient diffusion hypothesis with a fixed turbulent Prandtl number based on empirical values from simple turbulent flows. However, in more complex configurations, this model is known to be inaccurate. A methodology has been developed which uses experimental data to determine optimal uniform anisotropic turbulent diffusivity values for an angled jet in cross-flow. This configuration has applications in film cooling for gas turbine blades. The measurements, obtained by magnetic resonance imaging techniques, provide 3D time-averaged velocity and concentration fields. The mean velocity field is fed into a Reynolds-Averaged Advection Diffusion solver, which uses a spatially-uniform anisotropic turbulent diffusivity model to solve for the mean coolant concentration distribution. This distribution can be compared to the experimentally-obtained concentration field by means of an error metric that quantifies the difference between the computational and experimental concentration fields. By minimizing this error, an optimal value of the anisotropic turbulent diffusivity is determined.

8:13AM M22.00002 Large-eddy simulation of short-range dispersion from localized sources in an urban-like canopy , DAVID PHILIPS, Stanford University, RICCARDO ROSSI, University of Bologna, GIANLUCA IACCARINO, Stanford University — Results from large-eddy simulation of passive scalar dispersion from point source releases in an urban-like canopy are presented. The canopy is comprised of a periodic array of variable height buildings with square cross-sectional areas. The buildings are immersed in a turbulent, pressure-driven flow with a roughness Reynolds number, $Re_\tau = 433$. Pressure gradient direction is varied between 0°, 45°, and 90° to examine the effects on dispersion when the prevailing wind encounters staggered, skewed, or aligned building arrangements, respectively. Additionally, source location is varied to assess the impact of local building geometry on plume development. Plume trajectories and growth rates for the various scenarios are examined. The vertical development of the plume is better characterized by the roughness parameterization of the canopy than the horizontal development which is more sensitive to local geometry.

8:26AM M22.00003 Small-scale statistics of passive scalars released from concentrated sources in turbulent channel flow[1] , LAURENT MYDLARSKI, EMMANUEL GERMAINE, LUCA CORTELEZZI, McGill University — In 2010, we presented complementary experimental and numerical results pertaining to the large-scale statistics of a turbulent passive scalar released downstream of a line source in fully-developed turbulent channel flow. Our latest results relate to the evolution of the scalar dissipation rate ($\varepsilon_\theta \equiv \alpha\langle(\partial\theta/\partial x_i)^2\rangle$) downstream of the line source, for two different wall-normal source locations. We present experimental and numerical PDFs of the 3 different temperature derivatives ($\partial\theta/\partial x_\gamma$), as well as the different components of ε_θ ($\varepsilon_{\theta_\gamma}$), and conditional expectations of $\varepsilon_{\theta_\gamma}$. We also examine the anisotropy of the components of $\varepsilon_{\theta_\gamma}$ and note that these can asymptote to an anisotropic final state. This is attributed to the presence of the mean velocity gradient in the channel, which induces an additional production term (in the wall-normal direction) in the ε_θ budget. However, it also appears that the degree of this final anisotropy decreases with increasing Reynolds number and proximity to the wall (the two being correlated).

[1]Funding graciously provided by NSERC (grants RGPIN 217169 and 217184)

8:39AM M22.00004 Local structure of scalar flux in turbulent passive scalar mixing , ADITYA KONDURI, DIEGO DONZIS, Texas A&M University — Understanding the properties of scalar flux is important in the study of turbulent mixing. Classical theories suggest that it mainly depends on the large scale structures in the flow. Recent studies suggest that the mean scalar flux reaches an asymptotic value at high Peclet numbers, independent of molecular transport properties of the fluid. A large DNS database of isotropic turbulence with passive scalars forced with a mean scalar gradient with resolution up to 4096^3, is used to explore the structure of scalar flux based on the local topology of the flow. It is found that regions of small velocity gradients, where dissipation and enstrophy are small, constitute the main contribution to scalar flux. On the other hand, regions of very small scalar gradient (and scalar dissipation) become less important to the scalar flux at high Reynolds numbers. The scaling of the scalar flux spectra is also investigated. The $k^{-7/3}$ scaling proposed by Lumley (1964) is observed at high Reynolds numbers, but collapse is not complete. A spectral bump similar to that in the velocity spectrum is observed close to dissipative scales. A number of features, including the height of the bump, appear to reach an asymptotic value at high Schmidt number.

8:52AM M22.00005 Universality of spectrum of passive scalar variance at very high Schmidt number in isotropic steady turbulence[1] , TOSHIYUKI GOTOH, Nagoya Institute of Technology — Spectrum of passive scalar variance at very high Schmidt number up to 1000 in isotropic steady turbulence has been studied by using very high resolution DNS. Gaussian random force and scalar source which are isotropic and white in time are applied at low wavenumber band. Since the Schmidt number is very large, the system was integrated for 72 large eddy turn over time for the system to forgot the initial state. It is found that the scalar spectrum attains the asymptotic k^{-1} spectrum in the viscous-convective range and the constant C_B is found to be 5.7 which is larger than 4.9 obtained by DNS under the uniform mean scalar gradient. Reasons for the difference are inferred as the Reynolds number effect, anisotropy, difference in the scalar injection, duration of time average, and the universality of the constant is discussed. The constant C_B is also compared with the prediction by the Lagrangian statistical theory for the passive scalar. The scalar spectrum in the far diffusive range is found to be exponential, which is consistent with the Kraichnan's spectrum. However, the Kraichnan spectrum was derived under the assumption that the velocity field is white in time, therefore theoretical explanation of the agreement needs to be explored.

[1]Grant-in-Aid for Scientific Research No. 21360082, Ministry of Education, Culture, Sports, Science and Technology of Japan

9:05AM M22.00006 Statistical and Geometrical Properties of the Scalar Gradient in Homogeneous Isotropic Turbulence , MICHAEL GAUDING, JENS HENRIK GOEBBERT, FABIAN HENNIG, NORBERT PETERS, RWTH Aachen University — The mixing of a passive scalar in statistically homogeneous isotropic turbulence is investigated. Here, the scalar gradient plays an important role, since production of small scales and smoothing down by molecular diffusion depend on it. The single-point probability density function (pdf) of the scalar gradient is characterized by long stretched exponential tails. We derive an equation for the probability density function of the scalar gradient from first principles. This equation is not closed due to the highly nonlocal and non-linear character of the equations of motion. We employ a statistical framework to simplify the unclosed terms which also provides insight into the mechanisms of scalar gradient production, diffusion, and dissipation. We further introduce a simple closure for the tails of the scalar gradient pdf. This closure can be motivated by special alignment properties of the scalar gradient with its dissipation tensor. The theory is validated by means of direct numerical simulations with various Schmidt and Reynolds numbers.

9:18AM M22.00007 ILES of Passive Scalar Mixing in Forced Isotropic Turbulence , ADAM WACHTOR, FERNANDO GRINSTEIN, Los Alamos National Laboratory, RICK DEVORE, Naval Research Laboratory, RAY RISTORCELLI, LEN MARGOLIN, Los Alamos National Laboratory — Predictability of scalar mixing by an under-resolved turbulent velocity field is investigated using ILES. Turbulent mixing of a passive scalar by forced, compressible, isotropic turbulence with a prescribed mean scalar gradient is studied. The simulation strategy uses a multi-dimensional FCT algorithm, with low wavenumber momentum forcing imposed separately for the solenoidal and dilatational velocity components. Effects of grid resolution on the flow and scalar mixing are investigated at turbulent Mach numbers 0.13 and 0.27. ILES captures the mixing transition as function of effective Reynolds number determined by grid resolution, including asymptotic behaviors and characteristic turbulent metrics.

9:31AM M22.00008 Passive scalar mixing in variable-density, buoyant turbulent flows , PHARES L. CARROLL, GUILLAUME BLANQUART, California Institute of Technology — The interplay between turbulence and buoyancy is not fully characterized despite its presence in a wide range of environmental phenomena and engineering problems. Although classical Kolmogorov theory states that the dissipative scales are purely isotropic, there is evidence that this no longer holds in the presence of buoyancy. In this a-priori analysis, we consider two incompressible, miscible fluids with different densities that are subject to external body forces (gravity). The simulation results are used to probe the effect of variable-density and buoyancy on turbulence generation, small-scale isotropy, kinetic energy evolution, and turbulent mixing. The presence of isotropic behavior at the Taylor micro- and dissipative scales is examined via the Favre Reynolds stress anisotropy tensor. Analysis is conducted on the alignment of vorticity with the direction of principle strains to verify observed directional preferences. The role of buoyancy in the generation of turbulence is isolated by examination of appropriate energy spectra. Finally, the efficacy of mixing at varying Atwood and Schmidt numbers is analyzed using the probability density function (PDF) of mixture-averaged specific volume, the PDF of the scalar dissipation rate, and the scalar energy spectra.

Tuesday, November 20, 2012 8:00AM - 10:10AM –

Session M23 Turbulence Theory: General II 30D - Chair: Duo Xu, Purdue University

8:00AM M23.00001 Experimental Test of Revised Similarity Hypotheses without Taylor's Hypothesis , JUN CHEN, DUO XU, School of Mechanical Engineering, Purdue University — Simultaneous velocity and scalar fields of a turbulent jet, measured by combined Particle Image Velocimetry (PIV) and Planar Laser Induced Fluorescence (PLIF), are used to test Refined Similarity Hypotheses (RSH) and its extension to passive scalar (RSH-P). Without introducing artificial effects introduced by Taylor's hypothesis in traditional single-point measurements, RSH is successfully validated in this study by direct examinations of its three hypotheses. However, RSH-P is partially supported, where the hypothesis of independent behavior of stochastic variable is not supported.

8:13AM M23.00002 An infinity of microscales for turbulence[1] , WILLIAM K. GEORGE, Imperial College of London — It is has long been accepted that the Kolmogorov microscale $\eta = (\nu^3/\varepsilon)^{1/4}$ is the smallest dynamically significant length scale of turbulence (e.g.,[1]), where ν is the kinematic viscosity and ε is the dissipation. Following George [2] it is argued that there are an infinity of smaller scales, say $\eta_n = (\nu^{n+3}/\varepsilon_n)^{1/(2n+4)}$ where ε_1 is the dissipation of the dissipation, ε_2 is the dissipation of the dissipation of the dissipation, etc. Each of these is equal to a spectral moment in homogeneous turbulence, $(2\nu)^{n+1}\int_0^\infty k^{2n+2}E(k)dk$. Time scales can be similarly defined. It is demonstrated how these play an important role, especially in non-stationary turbulence where Kolmogorov's equilibrium hypothesis is invalid.

[1] Tennekes and Lumley (1972) A First Course in Turbulence, MIT Press.
[2] George, W.K (2012) Asymp. Effect of Initial and Upstream Conditions on Turbulence, J. Fluids Engr, 134, 1061203-1–27.

[1]EU Marie Curie Program

8:26AM M23.00003 Experimental test of a missing spectral link in turbulence , PINAKI CHAKRABORTY, Okinawa Institute of Science and Technology, HAMID KELLAY, Univ. Bordeaux, France, TUAN TRAN, University of Twente, The Netherlands, WALTER GOLDBURG, University of Pittsburgh, NIGEL GOLDENFELD, University of Illinois, GUSTAVO GIOIA, Okinawa Institute of Science and Technology — Although the cardinal attribute of turbulence is the velocity fluctuations, these fluctuations have been ignored in theories of the frictional drag of turbulent flows. Our goal is to test a new theory that links the frictional drag to the spectral exponent α, a property of the velocity fluctuations in a flow. We use a soap-film channel wherein for the first time the value of α can be switched between 3 and 5/3, the two theoretically possible values in soap-film flows. Remarkably, the new theory holds in both soap-film flows and ordinary pipe flows, even though these types of flow are governed by different equations. We conclude that even where the governing equations are unknown and α can take anomalous values (as in sediment-laden rivers and polymer-doped oil pipelines), the frictional drag might be estimated from simple measurements of α.

8:39AM M23.00004 Linear stability analysis of homogeneous three-dimensional turbulent flows , ANAND MISHRA, SHARATH GIRIMAJI, Texas A&M University — We examine the stability characteristics of homogeneous three-dimensional mean flows. Such mean fields can be categorized based on the invariants of the velocity gradient tensor. In this study, the linear stability of different three-dimensional mean-flow topologies and the action of pressure in each category are investigated. Expressly, this entails an analysis of the Kelvin-Moffat system in Fourier space. The concomitant invariant sets and their appurtenant bifurcations are explicated. Thence, the stability characteristics of the system are analyzed, apropos individual modes (i.e., Hydrodynamic stability) and the statistical ensemble (Rapid Distortion Theory). Such understanding can lead to improved pressure-strain correlation models.

8:52AM M23.00005 Multiscale Characterisation of Helical Properties in Homogeneous Turbulence , WOUTER BOS, Ecole Centrale de Lyon, FRANK JACOBITZ, University of San Diego, KAI SCHNEIDER, Aix-Marseille Université, MARIE FARGE, Ecole Normale Supérieure — This study investigates the helical properties of five prototypical homogeneous turbulent flows: statistically steady forced isotropic turbulence, decaying isotropic turbulence, decaying rotating turbulence, growing sheared turbulence, and growing rotating sheared turbulence. A solenoidal uncorrelated Gaussian random field is included in the analysis as a sixth comparison case. The scale-dependent helical properties of the cases are studied using an orthogonal wavelet decomposition. It was observed that flows with growing turbulent kinetic energy and turbulent motion at large scales show a maximum in the velocity helicity probability distribution functions (PDFs) at zero, corresponding to a trend to local two-dimensionalization of the flow with vorticity and velocity being perpendicular. Flows with decaying turbulent kinetic energy and turbulent motion at small scales, however, show maxima of the velocity helicity PDFs at plus and minus one, indicating a preference for helical motion with alignment or anti-alignment of vorticity and velocity. Joint PDFs of relative velocity helicity and relative vorticity helicity show that the quantities tend to have the same sign for all flows including the random field, indicating that vorticity helicity dissipates velocity helicity.

9:05AM M23.00006 Measurements of Anisotropy in Turbulence using SO(3) decomposition , GREG VOTH, Wesleyan University, SUSANTHA WIJESINGHE, Wayamba University of Sri Lanka — We use SO(3) decomposition of 3D particle tracking measurements to study the anisotropy of turbulence in a flow between oscillating grids. SO(3) decomposition is a powerful tool for determining the anisotropy as a function of scale, but experimental measurements of 3D anisotropy have proven to be difficult. Barriers that have hindered previous efforts to make these measurements include contamination from anisotropic sampling and the large data sets required for convergence of higher order anisotropic sectors. We use a real-time image compression system to obtain very large data sets of high speed video and to detect and correct for anisotropic sampling. We measure scaling exponents in the anisotropic sectors of the longitudinal structure functions up to j=4. Our results are consistent with previous results from numerical simulations and hot wire anemometry indicating that the scaling exponents at all orders increase with increasing j, so the small scales approach isotropy. We also condition the SO(3) decomposed structure functions on the instantaneous state of the large scales which provides an alternative way to probe the decay of anisotropy. We find that although smaller scales are not becoming independent of the large scales, but they are becoming isotropic.

9:18AM M23.00007 Mean shear regulates the intermittency of energy dissipation rate , KHANDAKAR MORSHED, LAKSHMI DASI, Colorado State University — We studied the multi-fractal properties of the instantaneous fluctuations of the turbulent kinetic energy dissipation rate, ε in the strongly anisotropic flow past a backward facing step. Measurements correspond to time-resolved PIV at Reynolds number, Re= 13600, 9000, and 5500 based on the free stream velocity and step height. Results indicate a significant dependence of the intermittent dissipation rate signal with respect to Re and local mean shear, S. Probability analysis showed that the fluctuations in ε are less skewed around its mean in regions of intense shear. The frequency of relatively intense bursts of intermittent fluctuations in ε appear to be dependent on the magnitude of these events. Lacunarity, a measure that characterizes such magnitude and temporal scale dependent intermittency of fluctuating signals, revealed that intermittency in ε reduces with S across all temporal scales. However, the intermittency of ε appears to increase with burst magnitudes. We discuss the implications of these results on the established multi-fractal picture of small-scale turbulence and the effects of large scale anisotropy.

9:31AM M23.00008 Energy spectrum in the wavenumber-frequency domain from Kraichnan's random sweeping hypothesis with mean flow , MICHAEL WILCZEK, Institute for Theoretical Physics, Univ. Muenster, YASUHITO NARITA, Institut fuer Geophysik und extraterrestrische Physik, TU Braunschweig — The energy spectrum in the wavenumber-frequency domain for turbulent flows is derived based on Kraichnan's random sweeping hypothesis with additional mean flow. The resulting model spectrum is parametrized by two parameters, the mean flow velocity and the sweeping velocity associated with Doppler shift and Doppler broadening, respectively. Among others, it has the interesting property that the power-law index of the one-dimensional wavenumber spectrum translates to the frequency spectrum, invariant for arbitrary choices of mean and sweeping velocity. In this talk, various properties of the model including implications for single- and multi-point measurements of turbulent flows are discussed, and the relation to the recently introduced elliptic model for space-time correlations is highlighted.

9:44AM M23.00009 Scaling of the mean length of streamline segments in various turbulent flows , PHILIP SCHÄFER, MARKUS GAMPERT, JONAS BOSCHUNG, NORBERT PETERS, RWTH Aachen University — Streamlines constitute natural geometries in turbulent flow fields. The latter can be partitioned into segments based on the zero crossings of the gradient of the absolute value of the velocity field along the streamline. Streamline segments can further be characterized by the sign of the gradient of the absolute value into positive and negative ones. Then, most of the statistical properties of streamline segments are captured in the joint probability density function of the arclength between and the velocity difference at the ending points. An analysis based on a model equation for the length distribution of streamline segments and the characteristic size of extreme points of the absolute value of the velocity field along the streamline yields that the mean length of the latter should scale with the geometrical mean of the Kolmogorov microscale and the Taylor microscale. This theoretical prediction is confirmed based on four different direct numerical simulations of turbulent flow fields with Taylor based Reynolds numbers ranging from 50 – 300. The database consists of two homogeneous isotropic decaying and one forced field. Furthermore, the case of a homogeneous shear flow is investigated.

9:57AM M23.00010 Turbulence theory and infrared images falsify the 2011 Nobel Prize in Physics , CARL GIBSON, University of California at San Diego — Turbulence defined by the inertial vortex force explains Planck scale big bang processes as temporary, rendering a permanent Einstein cosmological constant Λ and a positive expansion rate of the universe driven by anti-gravitational dark energy forces unnecessary. Large kinematic viscosity stresses during the plasma epoch from 10^{11} s to 10^{13} s cause fragmentation by proto-super-cluster-voids at 10^{12} s and proto-galaxies at the 10^{13} s transition to gas. Fragmentation of gas proto-galaxies is at Earth-mass planet viscous scales in Jeans mass clumps of a trillion planets. These Proto-Globular-star-Clusters (PGCs) freeze to form the dark matter of galaxies according to the Gibson (1996) Hydro-Gravitational-Dynamics (HGD) theory, and as observed by Schild (1996) by quasar microlensing. White dwarf carbon stars explode as Supernovae Ia events (SNeIa) when their mass increases to 1.44 solar, providing the standard candles used to justify the Nobel Prize claim of a positive expansion rate. However, if all stars form from primordial planet mergers in PGC clumps as claimed by HGD cosmology, the SNeIa become subject to a systematic dimming error depending on the line of sight to the event. New space telescope infrared images strongly support HGD cosmology.

Tuesday, November 20, 2012 8:00AM - 9:57AM –

Session M24 Compressible Flows I 30E - Chair: Oleg Vasilyev, University of Colorado at Boulder

8:00AM M24.00001 A molecular dynamics simulation demonstrating the invalidity of the Navier-Stokes Fourier (NSF) equations for compressible gaseous continua at all Knudsen numbers , HOWARD BRENNER, Massachusetts Institute of Technology, NISHANTH DONGARI, JASON REESE, University of Strathclyde — While it is well known to experimental gas kineticists and other fluid mechanicians that the NSF equations are invalid for noncontinua (rarefied gases) owing to Knudsen number effects, it is nevertheless universally believed that the NSF equations are valid for gaseous continua, namely when the Knudsen number is vanishingly small. This assumption is shown by molecular dynamics simulations to be wrong. This is demonstrated by performing simulations for monatomic gaseous continua undergoing steady-state rigid-body rotation relative to an inertial observer in a rigid circular cylinder that is thermodynamically isolated from its surroundings. The NSF equations, which are universally believed to govern the outcome of this elementary experiment, predict that the temperature will be uniform throughout the gas. In fact, the results of the simulation show that the temperature actually increases radially outward from the center of the cylinder to the wall by a significant amount, the greater the cylinder's angular velocity the greater the amount. Whereas the NSF paradigm fails to predict this nonisothermal radial temperature variation, the temperature distribution is shown to be accurately predicted by the recently proposed bivelocity hydrodynamic paradigm.

8:13AM M24.00002 A comprehensive theory for the response of gases to localized, transient heat addition , DAVID R. KASSOY, Retired — The response of a gas to localized, transient heat addition depends upon the amount of energy added during the heating period and the ratio of the heating time scale, t(H) to the local acoustic time, t(A). When the ratio is small the process occurs at nearly constant volume conditions, pressure rises with temperature while the density decrease is small. The local expansion Mach number is small. Gas expelled from the boundary of the high-pressure hot spot is the source of mechanical waves in the unheated environmental gas. The range of responses includes acoustic waves, shocks and very strong blast waves. When the amount of energy added exceeds an explicit limit the heating process is fully compressible with a substantial internal Mach number. When the time scale ratio is large energy addition to the volume leads to a nearly constant pressure process with the density inversely proportional to the rising temperature. The local expansion Mach number will range widely, depending on the amount of energy added and the size of the now large time scale ratio. Finally, a systematic formulation for the acoustic response of a gas, confined in a rectangle, to modest spatially distributed transient energy addition on a heating time scale t(H)=O(t(A)) is described.

8:26AM M24.00003 Delta-Measure Perturbations of a Contact Discontinuity , ROY BATY, Los Alamos National Laboratory — In this presentation, nonstandard analysis is applied to study generalized function perturbations of contact discontinuities in compressible, inviscid fluids. Nonstandard analysis is an area of modern mathematics that studies extensions of the real number system to nonstandard number systems that contain infinitely large and infinitely small numbers. Perturbations of a contact discontinuity are considered that represent one-dimensional analogs of the two-dimensional perturbations observed in the initial evolution of a Richtmyer-Meshkov instability on a density interface. Nonstandard predistributions of the Dirac delta measure and its derivatives are applied as the perturbations of a contact discontinuity. The one-dimensional Euler equations are used to model the flow field of a fluid containing a perturbed density interface and generalized solutions are constructed for the perturbed flow field.

8:39AM M24.00004 Effect of Large Bulk Viscosity on Two-Dimensional Transonic Flow[1] , MARK CRAMER, Virginia Polytechnic Institute and State University — We examine steady two-dimensional transonic flows over a thin airfoil or turbine blade. The wing Reynolds number is taken to be large and the fluid is described by the classical Navier-Stokes equations. The bulk viscosity is taken to be large compared to the shear viscosity. We use the Method of Matched Asymptotic Expansions to give the conditions under which the effects of large bulk viscosity are no longer negligible. We show that longitudinal viscous effects must be considered at lowest order when the ratio of bulk to shear viscosity is on the order of the product of the conventional Reynolds number times the two-thirds power of the non-dimensional airfoil thickness. Under these conditions the flow is shown to be frictional, irrotational, and governed by the viscous form of the transonic small disturbance equation.

[1]This work was supported by NSF Grant CBET-0625015.

8:52AM M24.00005 Effect of Large Bulk Viscosity on High-Speed Separation[1] , FATEMEH BAHMANI, MARK CRAMER, Virginia Polytechnic Institute and State University — We examine the effect of large bulk viscosity on the classical problem of two-dimensional shock boundary layer interaction. The flow is taken to be steady, supersonic and the plate is taken to be flat and adiabatic. The boundary layer is taken to be laminar and the fluid is modeled as a perfect gas with a bulk viscosity which is large compared to its shear viscosity. The flow details are computed using a fifth order weighted essentially non-oscillatory (WENO) finite difference scheme and 3rd order Runge Kutta scheme for the spatial and temporal discretizations. The primary result of interest is the suppression of separation when the fluid has a bulk viscosity which is large compared to the shear viscosity.

[1]This research is supported by NSF grant CBET-0625015.

9:05AM M24.00006 Modeling of the subgrid scale viscous/scalar dissipation in compressible turbulence , NAVID S. VAGHEFI, SUNY at Buffalo, MEHDI B. NIK, PATRICK PISCIUNERI, PEYMAN GIVI, University of Pittsburgh, CYRUS K. MADNIA, SUNY at Buffalo — Results are presented of subgrid scale (SGS) viscous/scalar dissipation models using $a\ priori$ analysis of compressible turbulent flows. This is done via assessment of DNS of several turbulent flow configurations at varying compressibility levels, Reynolds and Schmidt numbers. These models will be used as sub-closures in the LES via FDF of compressible turbulence. Optimum model parameters are calculated by maximizing the correlation coefficients between the SGS exact and modeled terms, and optimal estimators are used to verify the results. The effects of the filter width are also assessed for sub-closures. Different methods for calculating the model coefficients are evaluated and it is shown that a dynamic procedure based on the global SGS equilibrium between the production and dissipation produces the best results.

9:18AM M24.00007 A Characteristic-Based Volume Penalization Method for Compressible Viscous Flows in Complex Geometries[1] , ERIC BROWN-DYMKOSKI, NURLYBEK KASIMOV, OLEG VASILYEV, University of Colorado at Boulder — This is the first of two talks on new volume penalization method for numerical simulations of compressible flows around solid obstacles of complex geometries. This approach overcomes two major limitations of Brinkman penalization – the inability to model Neumann boundary conditions and shock reflection of solid boundaries. Boundary conditions on the fluxes are achieved through characteristic propagation into the thin layer inside of the obstacles. Inward pointing characteristics ensure nonphysical solution inside the obstacle does not propagate out to the fluid. The Dirichlet boundary conditions are enforced similarly to Brinkman penalization. Parameters defining the penalization terms are chosen so that they act on a much faster timescale than the characteristic time scale of the flow. A principle advantage of this method is the parameters provide a systematic means of controlling the error. The new approach is general and applicable to wide variety of flow regimes. This talk focuses on the application of the method to the Navier-Stokes equations. It is rigorously shown that the solution of the penalized problem converges towards the exact solution with the convergence of the penalization parameters. Examples of application to compressible viscous flows are given and discussed.

[1]This work was supported by ONR MURI on Soil Blast Modeling.

9:31AM M24.00008 A Characteristic-Based Volume Penalization Method for Compressible Inviscid Flows in Complex Geometries[1] , NURLYBEK KASIMOV, ERIC BROWN-DYMKOSKI, OLEG VASILYEV, University of Colorado at Boulder — This is the second of two talks on new volume penalization method for numerical simulations of compressible flows around solid obstacles of complex geometries. This approach overcomes two major limitations of Brinkman penalization – inability to model Neumann boundary conditions and shock reflection of solid boundaries. Boundary conditions on the fluxes are achieved through characteristic propagation into the thin layer inside of the obstacles. Inward pointing characteristics ensure nonphysical solution inside the obstacle does not propagate out to the fluid. Dirichlet boundary conditions are enforced similarly to Brinkman penalization. Parameters defining the penalization terms are chosen so they act on a much faster timescale than the characteristic time scale of the flow. A principle advantage of this method is parameters provide a systematic means of controlling the error. New approach is general and applicable to a wide variety of flow regimes. This talk focuses on the application of the method to the Euler equations. The main difference compared to Navier-Stokes formulation is the handling of slip boundary conditions and the effect of the curvature in the momentum equation. Examples of supersonic compressible inviscid complex geometry flows are given and discussed.

[1]This work was supported by ONR MURI on Soil Blast Modeling.

9:44AM M24.00009 Numerical modeling of a compressible multiphase flow through a nozzle , URSZULA NIEDZIELSKA, JASON RABINOVITCH, GUILLAUME BLANQUART, California Institute of Technology — New thermodynamic cycles developed for more efficient low temperature resource utilization can increase the net power production from geothermal resources and sensible waste heat recovery by 20-40%, compared to the traditional organic Rankine cycle. These improved systems consist of a pump, a liquid heat exchanger, a two-phase turbine, and a condenser. The two-phase turbine is used to extract energy from a high speed multiphase fluid and consists of a nozzle and an axial impulse rotor. In order to model and optimize the fluid flow through this part of the system an analysis of two-phase flow through a specially designed convergent-divergent nozzle has to be conducted. To characterize the flow behavior, a quasi-one-dimensional steady-state model of the multiphase fluid flow through a nozzle has been constructed. A numerical code capturing dense compressible multiphase flow under subsonic and supersonic conditions and the coupling between both liquid and gas phases has been developed. The output of the code delivers data vital for the performance optimization of the two-phase nozzle.

Tuesday, November 20, 2012 8:00AM - 10:10AM –

Session M25 Flow Control: External flows 31A - Chair: Thomas Corke, University of Notre Dame

8:00AM M25.00001 Distributed forcing of the flow past a blunt-based axisymmetric bluff body[1] , THIERRY JARDIN, YANNICK BURY, ISAE, DAEP TEAM — The topology of bluff body wakes may be highly sensitive to forcing at frequencies close to intrinsic flow instabilities. In a similar way, a steady but spatially varying forcing at wavelengths close to specific flow instabilities can lead to analogous outcomes. Such forcing is commonly referred to as distributed forcing. However, although distributed forcing has proven to be a relevant control strategy for three-dimensional flows past nominally two-dimensional geometries (e.g. extruded circular cylinder at Re $>$ 180), its impact on the flow past nominally three-dimensional geometries is still unknown. Here we assess the receptivity of the flow past a blunt-based axisymmetric bluff body to an azimuthally distributed forcing applied at the periphery of the bluff-body base. We show that the impact of RSPa, RSPb and RSPc instabilities on the drag fluctuations experienced by the bluff body can be suppressed, depending on the forcing wavelengths.

[1]The authors acknowledge the French Ministry of Defence and DGA for funding this work.

8:13AM M25.00002 Bio-Inspired Pressure Sensing for Active Yaw Control of Underwater Vehicles , AMY GAO, MICHAEL TRIANTAFYLLOU, Massachusetts Institute of Technology — A towed underwater vehicle equipped with a bio-inspired artificial lateral line was constructed and tested with the goal of active detection and correction of the vehicle's angle of attack. Preliminary experiments demonstrate that a low number of sensors are sufficient to enable the discrimination between different orientations, and that a basic proportional controller is capable of keeping the vehicle aligned with the direction of flow. We propose that a model based controller could be developed to improve system response. Toward this, we derive a vehicle model based on a first-order 3D Rankine Source Panel Method, which is shown to be competent in estimating the pressure field in the region of interest during motion at constant angles of attack, and during execution of dynamic maneuvers. To solve the inverse problem of estimating the vehicle orientation given specific pressure measurements, an Unscented Kalman Filter is developed around the model. It is shown to provide a close estimation of the vehicle state using experimentally collected pressure measurements. This demonstrates that an artificial lateral line is a promising technology for dynamically mediating the angle of a body relative to the oncoming flow.

8:26AM M25.00003 Large eddy simulation of the flow past a twisted cylinder[1] , JAE HWAN JUNG, Department of Naval Architecture and Ocean Engineering, Pusan National University, HYUN SIK YOON, Global Core Research Center for Ships and Offshore Plants, Pusan National University, CHANG YOUNG CHOI, School of Mechanical Engineering, Pusan National University — Large eddy simulation of flow past a twisted cylinder has been carried out at a Reynolds number of 300 based on the cylinder diameter and the free stream velocity using finite volume method. The twisted cylinder has been formed by rotating the elliptic cross sectional area along the spanwise direction. For an ellipse, different eccentricities are considered to observe the effect of eccentricity on the flow fields. The excellent comparisons with previous studies for the cases of a smooth cylinder and a wavy cylinder having sinusoidal variation in cross sectional area along the spanwise direction guarantee the accuracy of present numerical methods. The instantaneous vortical structures in wake of the twisted cylinder are compared with those of the circular and wavy cylinders. In general, the shear layer of the twisted cylinder covering the recirculation region is more elongated than those of the circular and the wavy cylinders. Successively, vortex shedding of the twisted cylinder was considerably suppressed, compared with those of the circular and wavy cylinders. Consequently, the twisted cylinder achieved a large amount of the drag reduction and especially the significant suppression of the fluctuating lift coefficient.

[1]This work was supported by the National Research Foundation of Korea (NRF) grant funded by the Korea government (MEST) through GCRC-SOP (No. 2011-0030662).

8:39AM M25.00004 Flow Separation Control with Rotating Cylinders , JAMES SCHULMEISTER, MIT, JASON DAHL, University of Rhode Island, GABRIEL WEYMOUTH, Singapore-MIT Alliance for Research and Technology, MICHAEL TRIANTAFYLLOU, MIT — The use of small counter-rotating rotating cylinders to control flow separation and reduce the drag of a fixed circular cylinder in cross-flow is investigated experimentally at Reynolds number (Re) 52,000 and computationally at Re 100 and 10,000. The moving surface of the control cylinders imparts momentum to the flow near the location of flow separation. The transfer of momentum delays separation further downstream and thereby reduces drag. The relationship between drag and rotation rate is found to be Reynolds number regime dependent; at Re = 100 the drag decreases linearly with rotation rate and at Re = 10,000, the relationship is non-linear. This non-linearity appears to be due to the interaction between vortex shedding from the small control cylinders (which does not occur at Re 100) and the main cylinder wake. Finally, the power consumed by the active control mechanism is considered and estimated to be significantly smaller than the power savings in reduced drag.

8:52AM M25.00005 Optimization of Feedback Control of Flow over a Circular Cylinder[1] , DONGGUN SON, EUIYOUNG KIM, HAECHEON CHOI, Seoul National University — We perform a feedback gain optimization of the proportional-integral-differential (PID) control for flow over a circular cylinder at $Re = 60$ and 100. We measure the transverse velocity at a centerline location in the wake as a sensing variable and provide blowing and suction at the upper and lower slots on the cylinder surface as an actuation. The cost function to minimize is defined as the mean square of the sensing variable, and the PID control gains are optimized by iterative feedback tuning method which is a typical model free gain optimization method. In this method, the control gains are iteratively updated by the gradient of cost function until the control system satisfies a certain stopping criteria. The PID control with optimal control gains successfully reduces the velocity fluctuations at the sensing location and attenuates (or annihilates) vortex shedding in the wake, resulting in the reduction in the mean drag and lift fluctuations.

[1]Supported by the NRF Program (2011-0028032)

9:05AM M25.00006 Control of the Shock-Induced Flow Separation over Convex Surfaces , ABRAHAM N. GISSEN, BOJAN VUKASINOVIC, ARI GLEZER, Georgia Tech — The present experimental investigation focuses on shock-induced flow separation off a convex surface geometry under subsonic upstream channel flow. In particular, the emphasis is placed on the narrow pre-chocked regime (M $\approx$ 0.6) with a signature localized shock that induces the flow separation off the surface. Both uncontrolled and controlled flows are studied by characterization of both the shock dynamics and the resulting separated flow over the convex surfaces of varying curvatures. The diagnostics tools include static and dynamic surface pressure measurements, assisted by qualitative and quantitative characterizations of the separated flow. Alteration of the shock wave dynamics and its coupling to the boundary layer separation and shear layer dynamics is achieved by active generation of streamwise vorticity. The active control is aimed at direct manipulation of either the shock dynamics or the flow separation. The former demonstrates control authority on the shock dynamics and its coupling to the shear layer, while the latter assesses both upstream and downstream coupling of the flow separation delay with the shock and shear layer, respectively.

9:18AM M25.00007 Aero-thermal optimization of film cooling flow parameters on the suction surface of a high pressure turbine blade , CAROLE EL AYOUBI, IBRAHIM HASSAN, WAHID GHALY, Concordia University — This paper aims to optimize film coolant flow parameters on the suction surface of a high-pressure gas turbine blade in order to obtain an optimum compromise between a superior cooling performance and a minimum aerodynamic penalty. An optimization algorithm coupled with three-dimensional Reynolds-averaged Navier Stokes analysis is used to determine the optimum film cooling configuration. The VKI blade with two staggered rows of axially oriented, conically flared, film cooling holes on its suction surface is considered. Two design variables are selected; the coolant to mainstream temperature ratio and total pressure ratio. The optimization objective consists of maximizing the spatially averaged film cooling effectiveness and minimizing the aerodynamic penalty produced by film cooling. The effect of varying the coolant flow parameters on the film cooling effectiveness and the aerodynamic loss is analyzed using an optimization method and three dimensional steady CFD simulations. The optimization process consists of a genetic algorithm and a response surface approximation of the artificial neural network type to provide low-fidelity predictions of the objective function. The CFD simulations are performed using the commercial software CFX. The numerical predictions of the aero-thermal performance is validated against a well-established experimental database.

9:31AM M25.00008 Numerical Investigation of Active Flow Control on Wind Turbines under Yaw Misalignment[1] , STEVEN TRAN, MANE, RPI, DAVID CORSON, Altair Engineering Inc, ONKAR SAHNI, MANE, RPI — Yaw misalignment dramatically increases unsteady aerodynamic loading on wind turbine blades over each revolution. The resulting fluctuating loads on each blade cause fatigue in the system and subsequently, failure leading to increased maintenance costs and unnecessary downtime. In this study we numerically analyze the effects of yaw misalignment on complete rotating wind turbines with blades of O(5m) in length. We consider two wind speeds at rated and above-rated regimes, where the effect of yaw misalignment is more pronounced. For the baseline configuration comparisons are made with the existing experimental data. To mitigate the resulting unsteady aerodynamic loading, we apply synthetic-jet based fluidic actuation in order to achieve fast-time response (in contrast to traditional yaw control strategies). O(5-10) jets are placed along the outer half of blade span. Along the chord two jet locations (x/c = 0.05 and 0.40) are considered. Actuation strategies for jets are based on partial loop control with pulse modulation. All simulations are based on unsteady Reynolds-averaged Navier-Stokes (URANS) equations.

[1]NYSERDA

9:44AM M25.00009 Feasibility Study of Using Gurney Flaps for Flow Control of Wind Turbine Blades[1] , POURYA NIKOUEEYAN, ANDREW MAGSTADT, JOHN STRIKE, JONATHAN NAUGHTON, University of Wyoming — Unsteady wind turbine aerodynamics due to atmospheric unsteadiness and rotation of the blade through a shear layer are phenomena that exceed the rate at which conventional blade pitch control mechanisms operate. Depending on the location on the blade, these rapidly varying effects can cause reduced aerodynamic efficiency, stall, unwanted oscillatory loads, and accompanying deflections. In this study, actively controlled Gurney flaps are investigated as a practical solution for alleviating these effects. Because of recent growth of the use of flatback airfoils in the root section of wind turbine blades, a DU97-W-300 derived flatback airfoil has been used in this study. The effect of flap height on lift and moment in static and dynamic conditions has been investigated by means of time-resolved pressure measurements. Static results show $\pm$30% changes in the section lift coefficient C_l with the Gurney flap, indicating sufficient authority for active control. An aeroelastic model based on the flatback airfoil geometry and correlations derived from the experimental Gurney flap results has been developed. Using a feedback control algorithm, the model indicates that a step disturbance can be quickly damped using closed-loop control of the Gurney flap.

[1]Support from DOE is acknowledged.

9:57AM M25.00010 Creation and Optimization of Compliant Flow for an Existing Wind Turbine Rotor Geometry , THEODORE WILLIAMS, THOMAS CORKE, JOHN COONEY, University of Notre Dame — A compliant flow is created on a wind turbine rotor through geometric optimization in order to make it more susceptible to active flow control. Feasible designs were limited to ones that can be implemented without permanent modification to the existing geometry. Computational fluid dynamics and quantitative optimization methods are employed to evaluate different design families that incorporate plasma flow control. Designs that resulted in the largest left control authority are presented. The application of these active lift designs on a horizontal wind turbine is discussed.

Tuesday, November 20, 2012 8:00AM - 10:10AM –

Session M26 Reactive Flows VII 31B - Chair: Antonio L. Snchez, University Carlos III de Madrid

8:00AM M26.00001 Rate Ratio Asymptotic Analysis of the Influence of Hydrogen on the Structure and Mechanisms of Extinction of Methane Flames in Laminar Nonpremixed Flows , KALYANASUNDARAM SESHADRI, Department of Mechanical and Aerospace Engineering, University of California at San Diego, XUE-SONG BAI, Division of Fluid Mechanics, Department of Energy Sciences, Lund University, Sweden, FORMAN WILLIAMS, Department of Mechanical and Aerospace Engineering, University of California at San Diego — Rate-ratio asymptotic analysis is carried out to elucidate the influence of hydrogen on the structure and mechanisms of extinction of methane flames in laminar nonpremixed flows. Steady, axisymmetric, laminar flow of two counter-flowing streams toward a stagnation plane is considered. One stream is made up of a mixture of methane, hydrogen, and nitrogen. The other stream is made up of a mixture of oxygen, and nitrogen. A reduced four-step chemical kinetic mechanism is employed. Chemical reactions are presumed to take place in a thin reaction zone that is established in the vicinity of the stagnation plane. On either side of this thin reaction zone, the flow field is inert. These inert regions are called the outer structure. The outer structure is analyzed first. It gives the matching conditions that is required in the analysis of the reaction zone. In the thin reaction zone chemical reactions are presumed to take place in two layers–an inner layer and an oxidation layer. In the inner layer methane is consumed and hydrogen and carbon monoxide are formed. In the oxidation layer oxygen, carbon monoxide and hydrogen are consumed. Critical conditions of extinction are predicted and are compared with experimental data and with results of numerical computation.

8:13AM M26.00002 Explosions in uncertain hydrogen-oxygen mixtures , JAVIER URZAY, NICOLAS KSEIB, Center for Turbulence Research, Stanford University, DAVID F. DAVIDSON, High-Temperature Gasdynamics Laboratory, Stanford University, GIANLUCA IACCARINO, Center for Turbulence Research, Stanford University, RON K. HANSON, High-Temperature Gasdynamics Laboratory, Stanford University — Uncontrolled residuals abound in combustors as a result of complex chemistry. The question to answer here is: How can we give a measure of the explosive tendency of a gaseous mixture when the initial composition is not known with absolute certainty? This study addresses the influences of uncertain amounts of residual radical impurities, namely H, O, OH and HO2 radicals, on the calculation and experimental determination of autoignition times in H2-O2 mixtures. To illustrate this point, shock-tube data is obtained in which the presence of residual radicals is evidenced by i) the detection of trace amounts of OH radicals in initial oxygen-argon mixtures and ii) the need of shortening the autoignition times calculated after integrations of the conservation equations when matching with experimental kinetics data. Regime diagrams of autoignition catalyzed by impurities above and below crossover are proposed, thereby summarizing the potential effects of residual dirt in shock-tube experiments and calculations. An experimentally-inspired model based on Bayesian inference is proposed for the uncertainty in the H-atom impurities in shock tubes. Monte-Carlo calculations of the conservation equations are performed using this model to assess the induced variabilities in the autoignition time.

8:26AM M26.00003 The simulation of a hydrogen-bubble reaction due to shock ignition , TEMISTOCLE GRENGA, SAMUEL PAOLUCCI, University of Notre Dame — We simulate the combustion of a hydrogen bubble in air ignited by a shock wave. The three dimensional compressible model includes detailed chemical kinetics, multi-component diffusion, Soret and Dufour effects, and state dependent transport properties. The reaction mechanism involves 9 species and 19 reversible reactions. The possibility of using a reduced chemical kinetics mechanism obtained through the G-Scheme is also explored. Results are compared with other numerical and experimental studies. The simulation is challenging since the physical and chemical phenomena lead to a large multiscale problem, which we solve using the parallel Wavelet Adaptive Multiresolution Representation (pWAMR) method. The method exhibits an impressive compression of the solution when compared to other methods. The algorithm is parallelized using a domain decomposition approach based on a Hilbert space-filling curve. pWAMR is able to capture all structures of O(μm) required for an accurate solution. The method is able to capture all scales using a relatively small number of degrees of freedom by adapting refinements to local demands of the solution. In addition, since the amplitudes of the wavelet transform provide a direct measure of the local error, we are able to produce a verified solution.

8:39AM M26.00004 Numerical study of a jet-in-hot-coflow burner with hydrogen-addition using the Flamelet Generated Manifolds technique[1] , SEYED EBRAHIM ABTAHIZADEH, JEROEN VAN OIJEN, PHILIP DE GOEY, Eindhoven University of Technology — Recently Mild combustion is subjected to intensive research because of its unique ability to provide high efficiency and low pollutant combustion simultaneously in industrial heating processes. In most practical Mild combustion applications, a fuel jet is ignited due to recirculation of hot burned gases. The impact of burned gases on autoignition and flame stabilization has been studied in a laboratory jet-in-hot-coflow (JHC) burner. Results of this study help us to understand recent experimental observations of the Delft group (DJHC burner) in which Dutch Natural Gas (DNG) is mixed with various amounts of H_2. The main focus is on the modeling of autoignition in the DJHC burner by using the Flamelet Generated Manifolds (FGM) technique. In this technique, kinetic information is tabulated with a few controlling variables which results in a significant decrease in simulation time. The FGM tabulation has been performed using igniting laminar counterflow diffusion flames. Since H_2 is present in the fuel composition, it is essential to include preferential diffusion effects in the table due to the high diffusivity of H_2. Based on results, the FGM table is capable to reproduce the autoignition of hydrogen containing fuel predicted by detailed chemistry in 1D counterflow flames.

[1]The Authors gratefully acknowledge financial support of the Dutch Technology Foundation STW.

8:52AM M26.00005 Numerical analysis and simulation of diffusion-free ignition delay times of unreacted pockets , JONATHAN REGELE, Iowa State University — Volumes of unreacted fluid surrounded by combustion products are observed in Deflagration-to-Detonation Transition (DDT) and unstable cellular detonations. The presence of this unburned reactant is typical of mixtures with large activation energies. Several different scales are involved in the consumption of unreacted pockets including the autoignition, diffusion, and acoustic times. Transport effects can influence the consumption rate of reactant within these pockets. In particular diffusion plays a major role when activations energies are large. In contrast, it has been shown that diffusion can play a minor role when reactive mixtures have small to moderately large activation energies. The current work focuses on the limit when diffusion effects are negligible and examines the dependence of delay time on initial temperatures and sizes. It is demonstrated that the ignition delay time is a function of both the initial temperature and the volumetric dimension of the fluid. Furthermore, the ignition delay time lies on a continuum scale with the constant pressure and constant volume ignition delay times demarcating the limiting extremes.

9:05AM M26.00006 Numerical simulation of autoigniting flames , RAJAPANDIYAN ASAITHAMBI, KRISHNAN MAHESH, University of Minnesota — Autoignition is highly sensitive to temperature and mixing. 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 algorithm solves the total chemical and sensible energy equation and heat capacities of species are obtained from thermodynamic tables. Chemical mechanisms in the Chemkin format is parsed and source terms are automatically linearized allowing the ability to simulate multiple fuels with minimal effort. Validation of the algorithm is presented and results from autoigniting non-premixed flames in vitiated coflow with different fuels are discussed.

9:18AM M26.00007 Investigation of ignition dynamics in a mixing layer with a vortex , SHYAM MENON, GUILLAUME BLANQUART, California Institute of Technology — Numerical simulations are developed to study ignition dynamics in a non-premixed H2/air layer where mixing is aided by an embedded vortex. A similar reacting flow situation is encountered in many practical devices including internal combustion engines and supersonic combustion. The current work improves upon previous work by using tabulated chemistry to predict ignition dynamics, greatly reducing the computational requirements. The key results include the prediction of ignition delay time as a function of hot air temperature and vortex characteristics such as vortex strength, characteristic size and center location. The simulations will be used to explore different regimes of ignition previously observed by varying vortex Reynolds numbers and oxidizer temperatures.

9:31AM M26.00008 A Priori Analysis of an Unsteady Flamelet Formulation for Reacting Jet in Cross Flows , WAI LEE CHAN, MATTHIAS IHME, University of Michigan, Ann Arbor, HEMANTH KOLLA, JACQUELINE CHEN, Sandia National Laboratories — The jet in cross flow (JICF) configuration is a common fuel-injection strategy in practical combustion systems. Characterized by anisotropic and three-dimensional turbulent structures, enhanced mixing, and flame-dynamical processes, the modeling of reacting JICF-configurations imposes several challenges for large-eddy simulations. The objective of this study is to assess the feasibility of using an unsteady flamelet model (USFM) for the simulation of reacting JICFs. To this end, an a priori analysis of the USFM is conducted using a direct numerical simulation database, and higher-order terms in the flamelet equations, representing cross-dissipation and secondary heat-transfer processes, are quantified. It is shown that these higher-order expansion terms that are commonly neglected in one-dimensional flamelet models become of importance at the flame-base and lee-side of the JICF. Possible model-extensions are discussed to account for these secondary processes.

9:44AM M26.00009 Collisional-Radiative Kinetics in Monatomic Gases[1] , HAI LE, ANN KARAGOZIAN, University of California, Los Angeles — A detailed model of electronic excited states is essential in capturing all the nonequilibrium processes of a partially ionized plasma by means of collisional and radiative interactions. This collisional-radiative (CR) model allows us to consider deviations from equilibrium distribution of the internal states, and is now more commonly used in the study of plasma discharges. Prior studies by Kapper and Cambier[2] and Panesi et al.[3] suggest that this level of detail is needed for an accurate prediction of the flow field, and it is particularly relevant to plasma-combustion interactions. The required number of excited states needed to be included in the CR model is often prohibitively large due to the nonequilibrium condition of the plasma. The consequence is a large system of ODE's which needs to be solved at each time step. A reduced mechanism for the CR model can be attained by grouping the upper states of the atomic state distribution (ASDF) into a pseudo-level in which the population is characterized either by a uniform distribution or a Boltzmann distribution. This talk presents both detailed and reduced models for an ionizing shock in Argon.

[1]Supported by the US Air Force/ERC, Inc. under subcontract RS111738

[2]Kapper & Cambier *J. Appl. Phys.* 109, 113308, 2011

[3]Panesi, et al., *J. Thermo. Heat Trans.*, 25, 3, 2011

9:57AM M26.00010 A Multiphase, Multicomponent Model for Combustion in the Titanium-Boron System[1] , SUSHIL KUMAR, JOHN B. BDZIL, MOSHE MATALON, D. SCOTT STEWART, Mechanical Science and Engineering, University of Illiinois, Urbana, IL 61801 — Modeling combustion by reaction diffusion processes in initially, separated pure compounds in the condensed phase of metals, their oxides and intermetallics, is an important area of research in material science. We present a one-dimensional combustion model of the Titanium-Boron system, which is thermodynamically consistent. A simplified stoichiometric reaction mechanism is used in which liquid Titanium and liquid Boron reacts to form Titanium diBoride. An analytical form, suggested by Fried and Howard [1], is used to represent the equilibrium equation of state (EOS) for the solid and liquid phases of all three substances. This EOS produces results that gives and excellent fit to the known experimental data for the pure chemical phases. A multicomponent mixture EOS is created to account for the phase transitions and reaction between the individual species. We use numerical simulations to examine the results from conservation equations at constant pressure and compare them to experiments.

[1] Fried, Laurence E., and W. Michael Howard. "Explicit Gibbs Free Energy Equation of State Applied to the Carbon Phase Diagram." Physical Review B 61.13 (2000): 8734-743.

[1]Supported by Defense Threat Reduction Agency, DTRA HDTRA1-10-1-0020 and the Air Force Research Laboratory FA8651-05-1-0003

Tuesday, November 20, 2012 8:00AM - 9:44AM –

Session M27 Boundary-Layer Instability II 31C - Chair: Jill Klentzman, University of Arizona

8:00AM M27.00001 Receptivity of high-speed boundary layers with real gas effects[1] , JILL KLENTZMAN, ANATOLI TUMIN, University of Arizona — The receptivity of high speed boundary layers in chemical nonequilibrium is investigated. A method is developed for the multi-mode decomposition of boundary layer flows including real gas effects, and the receptivity problem with small disturbances introduced at the wall is examined. The solution of the linearized Navier-Stokes equations, within the parallel flow approximation, is expressed in the form of normal modes, and the resulting differential equations for the amplitude functions are discretized using fourth-order finite differences. This discretized system is then in the form of a generalized eigenvalue problem, which yields a straight-forward definition of the associated adjoint system. A biorthogonality condition is formulated based on the adjoint eigenvectors. Assuming that a complete system of eigenfunctions of the discrete and continuous spectra exists, the biorthogonality condition allows for the projection of a solution onto the discrete modes, which can be utilized in the analysis of DNS results.

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

8:13AM M27.00002 Statistical inverse analysis of supersonic boundary-layer transition , GENNARO SERINO, OLAF MARXEN, FABIO PINNA, von Karman Institute for Fluid Dynamics, PAUL CONSTANTINE, CATHERINE GORLE, GIANLUCA IACCARINO, Stanford University — In environments with low boundary-layer disturbance levels representative of free flight in the atmosphere, the laminar-turbulent transition process for vehicles moving at supersonic speeds is typically governed by the convective amplification of high-frequency disturbances. A valid statistical characterization of the disturbance spectrum upstream of the transition location is a pre-requisite for an accurate prediction of transition. Statistical inverse analysis offers the possibility to provide such a characterization of relevant disturbance spectra. Using measured streamwise distributions of heat-transfer coefficients in the transitional zone, an intermittency factor can be defined. The intermittency factor is fed into an inference algorithm based on the Markov chain Monte Carlo method. This approach is applied to infer two characteristic variables, amplitude and frequency, of the disturbance spectrum upstream of the transition location. It relies on the repeated solution of the forward problem, i.e., the computation of intermittency curve given a certain probability density function of the disturbances. The solution of the forward problem employs linear-stability theory in conjunction with a critical threshold amplitude for transition.

8:26AM M27.00003 Stability of hypersonic compression cones[1] , HELEN REED, JOSEPH KUEHL, EDUARDO PEREZ, TRAVIS KOCIAN, NICHOLAS OLIVIERO, Texas A&M University — Our activities focus on the identification and understanding of the second-mode instability for representative configurations in hypersonic flight. These include the Langley 93-10 flared cone and the Purdue compression cone, both at 0 degrees angle of attack at Mach 6. Through application of nonlinear parabolized stability equations (NPSE) and linear parabolized stability equations (PSE) to both geometries, it is concluded that mean-flow distortion tends to amplify frequencies less than the peak frequency and stabilize those greater by modifying the boundary-layer thickness. As initial disturbance amplitude is increased and/or a broad spectrum disturbance is introduced, direct numerical simulations (DNS) or NPSE appear to be the proper choices to model the evolution, and relative evolution, because these computational tools include these nonlinear effects (mean-flow distortion).

[1]Support from AFOSR/NASA National Center for Hypersonic Research in Laminar-Turbulent Transition through Grant FA9550-09-1-0341 is gratefully acknowledged. The authors also thank Pointwise, AeroSoft, and Texas Advanced Computing Center (TACC).

8:39AM M27.00004 Azimuthal hotwire measurements in a transitional boundary layer on a flared cone in a Mach 6 quiet wind tunnel[1] , JERROD HOFFERTH, WILLIAM SARIC, Texas A&M University — Hotwire measurements of second-mode instability waves and the early stages of nonlinear interaction are conducted on a sharp-tipped, 5°-half-angle flared cone at zero angle of attack in a low-disturbance Mach 6 wind tunnel at $Re = 10{\times}10^6$ m^{-1}. Profiles of mean and fluctuating mass flux are acquired at several axial stations along the cone with a bandwidth of over 300 kHz. Frequencies and relative amplitude growth of second-mode instability waves are characterized and compared with nonlinear parabolized stability (NPSE) computations. Additionally, an azimuthal probe-traversing mechanism is used to investigate the character of the nonlinear stages of transition occurring near the base of the cone. Recent Direct Numerical Simulations (DNS) of a sharp cone at Mach 6 have shown that a fundamental resonance (or Klebanoff-type) breakdown mechanism can arise in the late stages of transition, wherein a pair of oblique waves nonlinearly interacts with the dominant two-dimensional wave to create an azimuthal modulation in the form of Λ-vortex structures and streamwise streaks. The azimuthal measurements will identify periodicity qualitatively consistent with these computations and with "hot streaks" observed in temperature sensitive paints at Purdue.

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

8:52AM M27.00005 Surface roughness effects on a blunt hypersonic cone[1] , NICOLE SHARP, JERROD HOFFERTH, EDWARD WHITE, Texas A&M University — The mechanisms through which distributed surface roughness produces boundary-layer disturbances in hypersonic flow are poorly understood. Previous work by Reshotko (AIAA 2008-4294) 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 quasi-random distributed roughness. Hotwire anemometry in the low-disturbance Texas A&M Mach 6 Quiet Tunnel shows a slight growth of fluctuations as well as vertical offset due to surface roughness at a range of unit Reynolds numbers. Spectral measurements indicate that the model is subcritical with respect to second mode growth, and azimuthal measurements are used to examine the high- and low-speed streaks characteristic of transient growth of stationary disturbances.

[1]Support from the AFOSR/NASA National Center for Hypersonic Research in Laminar-Turbulent Transition through Grant FA9550-09-1-0341 is gratefully acknowledged.

9:05AM M27.00006 Characteristics of a streak disturbance induced by an isolated roughness element[1] , KYLE BADE, AHMED NAGUIB, Michigan State University — A detailed description of a streak disturbance introduced in a Blasius boundary layer by an isolated roughness element will be presented. This work is motivated by the desire to understand the dependence of the evolution/instability of streamwise-oriented streaks (which play a key role in bypass transition) on the method by which they are generated. The proper scaling of the streamwise evolution of the streak disturbance energy is examined. This expands upon established Re_k^2 scaling (White, et al., Physics of Fluids, 2005) of streak disturbances induced by spanwise-periodic roughness element arrays. Examining different roughness heights, k, and employing a method that accounts for the streamwise growth of the streak's wall-normal and spanwise scales, it is found that the streak energy density scales with $\mathrm{Re}_k^{7/3}$, in the case of an isolated roughness element. The data used in the analysis are acquired using hotwire anemometry throughout a three-dimensional domain located downstream of a single cylindrical roughness element. These measurements are complemented by smokewire visualizations, which capture clearly three distinct disturbance states, dependent upon roughness element height; namely, stable streaks, streaks with intermittent turbulent bursts, and turbulent disturbances. Correspondence is established between these states and the streamwise evolution of the streak energy and the cross-stream disturbance profiles.

[1]NSF Grant: CMMI 0932546

9:18AM M27.00007 ABSTRACT WITHDRAWN –

9:31AM M27.00008 Two dimensional roughness effects on hypersonic boundary layer instability , KAHEI DANNY FONG, XIAOWEN WANG, XIAOLIN ZHONG, University of California at Los Angeles — Numerical simulations of 2-D roughness effects on modal growth are conducted for a hypersonic boundary layer. Perturbations correspond to pure mode S & mode F at 100 kHz and a wall normal velocity pulse with a frequency spectrum of 1MHz are considered. The evolution of perturbation at different frequency along the streamwise direction with the effect of surface roughness is studied by FFT. Our results show the importance of the relation between roughness location and the synchronization point, where the synchronization point is the point where mode S and mode F have the same phase velocity and synchronizes with each other. Its location can be obtained from the linear stability theory. The results show that if roughness is placed upstream of the synchronization point, perturbation is amplified. The amplification rate depends strongly on roughness height. On the other hand, if roughness is placed close to or downstream of the synchronization point, perturbation is damped. Similar to amplification, the strength of damping depends strongly on roughness height. A tentative explanation is that roughness alters the mean flow profile (ex: sonic line, inflection point). We believe this can be a candidate to explain the roughness-delayed transition as some experiments have shown.

Tuesday, November 20, 2012 8:00AM - 9:57AM –

Session M28 Industrial Applications II 32A - Chair: Simon Song, Hanyang University

8:00AM M28.00001 Flow and Heat Transfer Characteristics of the Staggered Slotted Semi Cylinders in a Cross Flow Heat Exchangers , SEDAT YAYLA, Yuzuncu Yil University, SEYFETTIN BEYIN, Star Technical university, ALPARSLAN OZTEKIN, Lehigh University — Transient 3-D dimensional turbulent flow simulations are conducted to examine flow and heat transfer characteristics in inline and staggered slotted semi-cylinders placed in a rectangular cross sectioned fin tube heat exchanger. Both Reynolds averaged Navier's equation and Large Eddy simulations model are employed to conduct simulations using Fluent-ANSYS. Predictions of transient simulations are compared against the results of the PIV flow visualization observations at Reynolds number 1500 and 4000. Measured and predicted velocity and the vorticity field in the wake of cylinders agree well with each other at both Reynolds number. The effect of the angle between the slotted semi cylinders and the flow direction is investigated for various values of Reynolds number in both laminar and turbulent flow regimes. Transient nature of the three dimensional flow structures with flow separation, reattachment and vortices are characterized. The effects of the flow structure on the heat transfer characteristics are determined by calculating the heat transfer coefficient along the surface of the semi cylinders.

8:13AM M28.00002 High speed imaging in icing windtunnel tests , DENNIS DE PAUW, PERCIVAL GRAHAM, ALI DOLATABADI, Concordia University — The detailed visualization and behavior of a spray impinging on a hydrophilic, and superhydrophobic aerodynamic shape in isothermal room and icing conditions can provide deep understanding of in-flight icing. A superhydrophobic coating has a very low surface energy so it can be used to counteract the ice accumulation. It also reduces the adhesion strength of ice to the surface which ensures easier removal of the ice during flight. The focus of the experiments primarily lies on the fundamental study of multiple droplet, i.e. spray, impact on a NACA 0012 airfoil in room and icing conditions. Under such conditions, important icing features such as rivulets and runback flow are observed. This provides us with the basics of ice formation on an aerodynamic surface. The study also focuses on the comparison between aluminum and superhydrophobic surfaces for ice accumulation in conditions which approach flight conditions. All the experiments are carried out in a small scale icing windtunnel using high speed photography with frame rates ranging from five thousand to fifty thousand frames per second.

8:26AM M28.00003 Development of MEMS-based thermal flow sensors for high sensitivity and wide range of flow rate[1] , WOONG KANG, HAE MAN CHOI, YONG MOON CHOI, Korea Research Institute of Standards and Science — We have proposed and demonstrated a novel design of MEMS-based thermal mass flow sensor for high sensitivity and wide flow range. Thermal mass flow sensors are able to measure small amount of gas flow such as process control gas via heat transfer phenomena between heater and thermopiles. To understand characteristics of the correlation between sensing performance and geometry of sensor components like heater and thermopile, various designed models were fabricated by using MEMS technology considering manufacturing efficiency. A evanohm R alloy heater and chromel-constantan thermopiles were formed on a $Si_3N_4/SiO_2/Si_3N_4$ sandwich type membrane for thermal performance enhancement. Characteristics tests between flow rate, heat power and sensitivity for fabricated models were conducted in low pressure gas flow standard system of KRISS (Korea Research Institute of Standard and Science) with MFC (Mass Flow Controller). Finally, the optimum geometry based on the non-uniform distribution of heater and thermopiles was determined according to characteristics comparison of designed and fabricated models. The developed thermal mass flow sensor can be adopted for low range flow rate (0 – 200 sccm) and also high one (up to 10 SLM) with high sensitivity.

[1]This work was supported by the Korea Research Institute of Standards and Science under the project 'Establishment of National Physical Measurement Standards and Improvements of Calibration/Measurement Capability,' grant 12011002

8:39AM M28.00004 Experimental Investigation of Flow and Thermal Patterns in the Rotated Arc Mixer[1] , OZGE BASKAN, MICHEL SPEETJENS, Eindhoven University of Technology, GUY METCALFE, Commonwealth Scientific and Industrial Research Organisation, HERMAN CLERCX, Eindhoven University of Technology — Thermal patterns emerging during the downstream evolution of temperature fields in industrial inline mixers have been studied numerically yet experimental observation remains outstanding. This research concerns a comparative analysis between experimental and numerical studies on the evolution of the temperature fields of a representative configuration, namely the Rotated Arc Mixer (RAM), and its correlation with the flow field. The RAM is an inline mixer that is composed of a stationary inner cylinder with consecutive 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 setup consists of a circular test section with apertures on the circumference and motor-driven belts imitating the rotating cylinder. Constant circumferential temperature is achieved by an enclosing annular hot-water reservoir. The 2D flow and temperature fields are measured by 2D Particle-Imaging Velocimetry and Infrared Thermography. Preliminary results have exposed a clear correlation between temperature and flow fields: thermal patterns evolve in accordance with the time-periodic flow patterns and become persistent ultimately.

[1]The authors gratefully acknowledge the support by Dutch Technology Foundation STW.

8:52AM M28.00005 Novel Model to Predict Minimum Coating Thickness for High Speed Slot Coating , ILHOON JANG, SIMON SONG, Department of Mechanical Engineering, Hanyang Univ., Korea — Recently slot coating is often applied to printed electronics for a flat display and in battery industry due to advantages such as the fast production rate and cost effectiveness. The accurate prediction of minimum coating thickness, closely related to coating stability, is a key issue in slot coating. It is because trial-and-error should be minimized when determining operating conditions of slot coating of which inks with metallic nano-particles are very expensive. So far, the viscocapillay model is known to provide good physical insight in a range of a low or moderate coating speed. However, its predictions are inaccurate for high coating speed since it doesn't consider the inertia of the ink flow arising at the high speed coating. In this study, we propose a novel model which accounts for the inertial effects. We performed detailed numerical analysis on ink flows of a slot coating to find out the cause of inaccurate prediction at a high speed coating and minimum coating thicknesses under various operating conditions. We found that the novel model prediction and numerical results are in excellent agreement in a wide coating speed range and that the new model can be applicable to an operating Reynolds number of an order higher than the viscocapillary model.

9:05AM M28.00006 Macro analysis of the electro adsorption process in a capacitive demonization cell during water desalination at developing and fully developed concentration regimes , CARLOS RIOS PEREZ, ONUR DEMIRER, REBECCA CLIFTON, RACHEL NAYLOR, CARLOS HIDROVO, The University of Texas at Austin — Capacitive deionization has become a desalination technique of large interest because of its added capability of energy recovery during the regeneration of the adsorbing electrodes. As in any separation practice, adequate modeling of the mass transport mechanisms present in the salt extraction process is crucial for the adequate dimensioning of the desalination cell and selection of the operation parameters. In this regard, this paper presents a simplified one-dimensional model of the concentration variation within a capacitive deionization cell. This model was solved at two distinctive regimes: developing, and fully developed convective diffusion layer. These solutions were used to estimate the net electro-adsorption rates by comparing the predictive variation of the minimum ratio of outlet to inlet solution concentrations with the corresponding measured values at various flow rates. A very good agreement between anticipated and measured outlet solution concentration transients validated the model and methodology to estimate the adsorption rates. This good concurrence between model and experiments evidence the capability of the proposed model to accurately simulate the effects of electrode saturation on the net electro-adsorption rate. Finally, the model and methodology presented were tested with experiments using brackish water concentration solutions.

9:18AM M28.00007 Near-Critical CO2 Flow Measurements and Visualization[1] , FARZAN KAZEMIFAR, DIMITRIOS KYRITSIS, University of Illinois at Urbana-Champaign — Carbon dioxide capturing and sequestration is one of the proposed solutions for reducing greenhouse gas emission. This technique will be used in big industrial plants with very high CO2 emissions. Handling such large flow rates requires high pressure and low temperature (in order to maximize density and minimize volumetric flow rate) which brings us close to the critical point of CO2 at approximately 74 bar and 31°C. This necessitates studying near-critical CO2 flows. In our experiment setup CO2 is compressed to supercritical pressures using a hydraulic accumulator. Pressurized CO2 then flows through the test section, which is a 2-ft long stainless steel tube with ID = 0.084 in. The flow rate is controlled by a needle valve downstream of the test section and the mass flow rate is measured using a coriolis mass flow meter. Temperature and pressure are monitored using two K-type thermocouples and pressure transducers at the inlet and exit of the test section. The pressure difference across the pipe is measured separately using a differential pressure transducer. In another set of experiments, the aforementioned test section is replaced with an optically accessible test section. In this setup high-speed imaging is used to visualize the flow inside the test section. We studied the recorded data in order to identify distinct flow regimes based on pressure drop as a function of pressure, temperature and mass flow rate.

[1]Acknowledgements: International Institute for Carbon-Neutral Energy Research (I2CNER)

9:31AM M28.00008 Study of fluid parameters in high pressure descaling valves , PARAM ADHIKARI, Department of Mechanical & Industrial Engineering, Youngstown State University, Ohio, YOGENDRA PANTA, Department of Mechanical & Industrial Engineering, College of Science, Technology, Engineering and Mathematics, Youngstown State University, Ohio — Our work is focused on the high pressure valves used for descaling purposes in steel mills. A reverse flow operation was set in one of such valves due to piping constraints. Computational approaches are being utilized to understand the fluid phenomena at such high pressures. Though the valve geometry accounts for the complete fluid flow path, a study has been initiated from an axisymmetric model of the valve core. The highly energized fluid from the descaling pump sets off a static pressure of 5000 psi at the valve inlet. It is responsible for continuous fluid flow rate of up to 208 gpm for fully open position. A Shear Stress Transport turbulence model is utilized to study pressure at nearly closed position of the poppet part while Renormalization Group Turbulence model is compared with Shear Stress Transport turbulence model for full opening position. A very low pressure developed below the poppet seat suggests the onset of cavitation zones which may lead to leakage. A full 3D model is studied after a complete studies of fluid phenomenon in the axisymmetric geometry. Using ANSYS Fluent, a commercial CFD software package, the poppet valve assembly was processed for modeling, meshing and setting up of physical parameters. Computational results show the cavitation intensities higher at small openings than at larger openings which is further verified by literature research and currently comparing with experiments.

9:44AM M28.00009 Electrowetting climbing of inclined water surfaces[1] , JUNQI YUAN, SUNG KWON CHO, University of Pittsburgh — It is well known that some small insects easily climb the inclined air-water interfaces simply by distorting the interfaces. This presentation will talk about bio-mimicking of the insect-climbing principle by electrically controlling the contact angle (so called electrowetting) and thus distorting the adjacent interfaces. For an experimental proof, we fabricated a small floating object whose sidewalls were covered with electrowetting electrodes. In response to the external voltage applied to the electrodes, the surfaces of the sidewalls can be switched from hydrophobic to hydrophilic states, resulting in interface distortions. It is confirmed that the interface distortions generate lateral forces that allow the floating object to climb inclined water surfaces. Detailed results along with effects of many parameters will be presented and discussed.

[1]This work was in part supported by the NSF grant (ECCS-1029318)

Tuesday, November 20, 2012 8:00AM - 10:10AM –

Session M29 Chaos, Fractals, and Dynamical Systems II 32B - Chair: Tom Solomon, Bucknell University

8:00AM M29.00001 ABSTRACT WITHDRAWN –

8:13AM M29.00002 Dynamical behavior of flame front instability induced by radiative heat loss , HIROSHI GOTODA, TAKUYA IKAWA, KOSHIRO MAKI, TAKAYA MIYANO, Ritsumeikan University — Exploring complexities in flame front instability leading to flame extinction is of current interest in combustion physics and nonlinear science. We investigate the dynamical properties of the flame front instability induced by radiative heat loss using nonlinear forecasting. The flame front instability represents high-dimensional chaos generated via the period-doubling cascade process, while its short-term behavior is predictable using a local nonlinear predictor based on the Sugihara-May method (G. Sugihara, R. M. May, Nature 344, 734 (1990), H. Gotoda et al., Chaos 22, 033106, (2012)). The feasibility of a new approach based on short-term prediction is also discussed in this work from the practical viewpoint of combustion systems.

8:26AM M29.00003 The nonlinear behaviour of a ducted premixed flame[1] , KARTHIK KASHINATH, IAIN WAUGH, University of Cambridge, SANTOSH HEMCHANDRA, Indian Institute of Science, MATTHEW JUNIPER, University of Cambridge — Nonlinear thermoacoustic oscillations are one of the most challenging problems in premixed gas turbine engine combustors. We investigate the nonlinear thermoacoustic behaviour of a ducted premixed flame using time domain analyses of the fully coupled thermoacoustic system. Nonlinear time series analysis tools are used to analyse the complex oscillations that the system exhibits. The system shows periodic, quasi-periodic, frequency-locked and chaotic oscillations depending on the operating condition. While such behaviour has been observed in experiments (Kabiraj et. al., Chaos 22, 023129 (2012)), this is the first attempt at modelling this system using an approach that captures the details of the flame front dynamics. The bifurcations of the system for changes in operating conditions are studied for a few different control parameters such as the flame equivalence ratio, geometry of the system and mean flow velocity. Furthermore, a route to chaos is identified in this system.

[1]Supported by EPSRC and Rolls Royce

8:39AM M29.00004 Front propagation in steady cellular flows: A large-deviation approach[1] , ALEXANDRA TZELLA, JACQUES VANNESTE, School of Mathematics, University of Edinburgh, UK — We examine the speed of propagation of chemical fronts modelled by the Fisher–Kolmogorov–Petrovskii–Piskunov nonlinearity in steady cellular flows. A number of predictions have been previously derived assuming small molecular diffusivity (large Péclet number) and either very slow (small Damköhler number) or very fast (large Damköhler number) chemical reactions. Here, we employ the theory of large deviations to obtain a family of eigenvalue problems from whose solution the front speed is inferred. The matched-asymptotics solution of these eigenvalue problems in the limit of large Péclet number provides approximations for the front speed for a wide range of Damköhler numbers. Two distinguished regimes are identified; in both regimes the front speed is given by a non-trivial function of the Péclet and Damköhler numbers which we determine. Earlier results, characterised by power-law dependences on these numbers, are recovered as limiting cases. The theoretical results are illustrated by a number of numerical simulations.

[1]The authors acknowledge support from EPSRC grant EP/I028072/1

8:52AM M29.00005 Experimental studies of stationary reaction fronts in a chain of vortices[1] , CARLEEN BOYER, Bucknell University, TOM SOLOMON, Bucknell University — We present results of experiments studying the behavior of the excitable Belousov-Zhabotinsky (BZ) reaction in a chain of alternating vortices with an imposed uniform wind. Previous experiments[2] have shown that fronts in this system are pinned for a wide range of imposed wind speeds, propagating neither forward against the wind nor in the downwind direction. We explain this behavior with a recent theory[3] that proposes the existence of *burning invariant manifolds* (BIMs) that act as local barriers to front propagation. Fronts are pinned when a BIM or a combination of BIMs spans the width of the vortex chain, blocking the reaction front. We show experimental measurements of the shape of the pinned front for a range of different wind speeds, and compare these shapes to the BIMs calculated theoretically. We also consider the dependence of the front shape on the location of the initial trigger for the front.

[1]Supported by NSF Grants DMR-0703635, DMR-1004744, and and PHY-1156964.

[2]M.E. Schwartz and T.H. Solomon, Phys. Rev. Lett. **100**, 028302 (2008).

[3]J. Mahoney, D. Bargteil, M. Kingsbury, K. Mitchell and T. Solomon, Europhys. Lett. **98**, 44005 (2012).

9:05AM M29.00006 Burning invariant manifolds and pinning of reaction fronts in spatially-disordered fluid flows[1] , TOM SOLOMON, MAYA NAJARIAN[2], Bucknell University — We present experiments that test the ideas of *burning invariant manifolds* (BIMs) for propagating fronts in spatially-disordered fluid flows with an imposesd wind. The disordered flow is driven by a magnetohydrodynamic forcing technique, and there is a uniform wind imposed on the flow with the use of a translation stage. Reaction fronts are produced using the excitable Belousov-Zhabotinsky chemical reaction. For a wide range of wind speeds, a complicated stationary front forms, pinned to the underlying vortex flow, neither propagating forward against the wind nor being blown backwards. The shape of the front depends significantly on the magnitude of the imposed wind. We test the hypothesis that the shape of the stationary front is determined by a collection of overlapping BIMs that act as barriers against forward movement of the reaction front. The location of the BIMs are predicted by integrating a three-dimensional set of ordinary differential equations[3] that describes the dynamics of an element of an evolving reaction front in the fluid flow.

[1]Supported by NSF Grants DMR-0703635, DMR-1004744, and PHY-1156964.

[2]Current address: Middlebury College, Middlebury, VT

[3]J. Mahoney, D. Bargteil, M. Kingsbury, K. Mitchell and T. Solomon, Europhys. Lett. **98**, 44005 (2012).

9:18AM M29.00007 Pinning fronts in advection-reaction-diffusion systems: a dynamical systems approach[1] , KEVIN MITCHELL, JOHN MAHONEY, JOHN LI, University of California, Merced — Recent experiments have demonstrated the pinning of reaction-diffusion fronts in magnetohydrodynamically-forced vortex flows. Specifically, a magnetic stage moving beneath the fluid layer "captures," and then drags, a reaction-diffusion pattern, which remains pinned to the frame of the stage. Here, we use dynamical systems techniques to explain the sequence of bifurcations that leads from an unpinned to a pinned state, as well as bifurcations that change the topological structure of the pinning fronts. We also explain how different pinning behavior can coexist within the same fluid flow, and analyze the associated basins of attraction. Our analysis is based on the recent concept of "burning" invariant manifolds (BIMs); BIMs extend the invariant manifolds traditionally used in passive advection to the case of reaction-diffusion systems.

[1]Supported by the US National Science Foundation under grant PHY-0748828.

9:31AM M29.00008 An FTLE analysis for reaction-diffusion fronts in fluid flows[1] , JOHN MAHONEY, KEVIN MITCHELL, University of California, Merced — The theory of advective transport depends heavily on the elucidation of organizing structures within the fluid. In a time-independent or time-periodic flow, one can define invariant manifolds. In a time-aperiodic flow, one often employs the finite-time-lyapunov-exponent (FTLE) and Lagrangian coherent structures. It has been recently demonstrated that fronts, e.g. reaction-diffusion fronts, propagating in time-periodic flows can also depend on such organizing invariant manifolds. In this talk, we describe an FTLE analysis for propagating fronts in two-dimensional fluid flows. In particular, we employ a dimension reduction technique to the front system so that a two-dimensional FTLE approach is feasible.

[1]NSF PHY-0748828

9:44AM M29.00009 Uncertainty propagation using spectral methods and flow map composition , DIRK M. LUCHTENBURG, STEVEN L. BRUNTON, CLARENCE W. ROWLEY, Princeton University — Uncertainty quantification is becoming more widely used in a variety of applications: for instance, when analyzing oil spills, one wants to predict the extent of the contaminated region, but the velocity field is not known precisely. We propose an efficient method for computing the propagation of a probability density function (PDF) through the long-time, nonlinear flow map associated with an uncertain fluid velocity field. Uncertain initial conditions and parameters are both addressed. The method approximates the short-time flow map by a spectral basis and uses flow map composition to construct the long-time flow map. The short-time flow map is characterized by small stretching and folding of the associated trajectories and hence can be represented by a relatively low-order basis. The composition of these low-dimensional bases then accurately describe the uncertainty behavior for long times. We use sampling of the spectral representations to compute stochastic quanti- ties, such as the mean and variance. The method is applied to several numerical examples including the long-time advection of a distribution of particles through an uncertain velocity field.

9:57AM M29.00010 Noise-induced complexity in active nonlinear spatially extended systems , MARC PRADAS, SERAFIM KALLIADASIS, Department of Chemical Engineering, Imperial College London, DMITRI TSELUIKO, School of Mathematics, Loughborough University, DEMETRIOS T. PAPAGEORGIOU, GRIGORIOS A. PAVLIOTIS, Department of Mathematics, Imperial College London — We study noise-induced phenomena on spatially extended systems (SES) that are close to the instability onset. We consider a degenerate noise that is acting on the subspace of stable modes only, and by means of a multiple scale analysis for general noisy SES we obtain an amplitude equation for the dominant mode. This then allows us to analytically investigate the noise effects on the dominant dynamics of the system. We observe that several non-trivial scenarios are possible depending on the stable modes the noise is acting on, including noise-induced critical transitions, intermittency and stabilisation when the noise is acting on the first stable mode only; or a noise filtering process, i.e. the dominant mode is not affected at all by the stochastic forcing when it is acting on the second stable mode. Our analytical findings are exemplified with a model SES, the noisy Kuramoto-Sivashinsky equation which describes, amongst many other different physical settings, the dynamics of a thin-liquid film flowing over a topographical substrate. In all cases, very good agreement between the theoretical predictions and numerical experiments is observed.

Tuesday, November 20, 2012 8:00AM - 10:10AM –

Session M30 General Fluids II 33A - Chair: Peter Schmid, LadHyX, Ecole Polytechnique

8:00AM M30.00001 ABSTRACT WITHDRAWN –

8:13AM M30.00002 Wetting and partiually wetting rivulets: the role of Reynolds number and boundary conditions , PETER VOROBIEFF, NIMA FATHI, The University of New Mexico, VAKHTANG PUTKARADZE, University of Alberta, KEITH MERTENS, LeapMotion.com — The behavior of gravity-driven rivulets flowing down an inclined plane or confined between two vertical planes has attracted considerable recent attention. We present a study of several fluids with different wetting properties in both of these arrangements, and discuss the effects of changes in the boundary conditions and the flow rate (both in terms of average Reynolds number and variability). Our experimental arrangement allows to introduce 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. For the case of the flow between two vertical planes, one of these regimes manifested for a partially wetting stream exhibits particularly interesting and visually striking features.

8:26AM M30.00003 Measurements of longitudinal surface waves in a soluble surfactant solution[1] , N. WASHUTA, X. LIU, University of Maryland, G.M. KORENOWSKI, Rensselaer Polytechnic Institute, J.H. DUNCAN, University of Maryland — Longitudinal wave trains generated at a surfactant-laden air-water interface are studied experimentally. The experiments are performed in a glass tank that is 75 cm long, 17.8 cm wide, and 4.5 cm deep. Longitudinal waves are generated using a Teflon barrier that spans the width of the tank and oscillates horizontally in the long direction of the tank. The local instantaneous surfactant concentration is measured non-intrusively 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 an incident angle of 60 degrees. Due to nonlinear optical effects, the reflected beam contains light with wavelengths of both 532 nm and its second harmonic, 266 nm. The ratio of the intensity of the 266-nm light to the 532-nm light is proportional to the concentration of surfactant on the surface. By measuring the local surfactant concentration versus time at a number of distances from the oscillating barrier, the wavelengths of the longitudinal waves are determined. The relationship between the surface dynamic properties of the surfactant and the measured dispersion relationship of the longitudinal waves is discussed.

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

8:39AM M30.00004 Water wave metamaterials , PHILIPPE PETITJEANS, PMMH - ESPCI, CARMEN PALACIOS, PMMH - ESPCI / Imperial College London, AGNÈS MAUREL, Institut Langevin - ESPCI, VINCENT PAGNEUX, Laboratoire d'Acoustique de l'Universit'e du Maine (LAUM) — The phenomenon of water wave deviation in a bended wave-guide has been studied experimentally. We propose a theoretical analogy to electromagnetism, from which we derive the mathematical tools to design a water wave-deviator. To obtain the effect of metamaterial in the case of surface waves, one has to design a water-bed consisting of periodic layers of two different heights inclined with a specific angle with respect to the direction of propagation of waves. We designed and built (using rapid prototyping) deviators with progressively increasing angles of bending, and their homologue wave-guides with a flat bottom. The wave elevation was measured with good accuracy in time and in space by an optical method. Results show a good efficiency of the wave-deviator. The wavefront maintains its original inclination once the wave crosses the bend (in contrary to the wave-guide with a flat bottom), however departs from the predicted behavior as the wavefront advances. The analysis of harmonics shows a reduction of backwards reflection and a strong decrease in higher modes excitation after the bend. The results are optimistic and might open new possibilities; ultimately those regarding the cloaking of floating structures which could, in the future, be used for coastal protection.

8:52AM M30.00005 Bouncy Fluid Jets , NAVISH WADHWA[1], SUNGHWAN JUNG, Department of Engineering Science and Mechanics, Virginia Tech, PAVLOS VLACHOS, Department of Mechnaical Engineering, Virginia Tech — Contrary to intuition, free fluid jets can sometimes "bounce" off each other upon collision, due to an entrained air film that keeps them separated. So far, there have only been a few descriptive studies of bouncing jets, since the first recorded observation by Rayleigh more than a century ago. We present a quantitative investigation of non-coalescence in jets of same fluid upon an oblique collision. Using a simple experimental set-up, we carried out a parametric study of the bouncing jets by varying the jet diameter, velocity, angle of inclination and fluid viscosity. Our results reveal a scaling law for the contact time of bouncing jets. We further investigate the transition of colliding jets from non-coalescence to coalescence, which seems to be caused by instability of the fluid interface. A dimensionless parameter, which is a function of the Normal Weber Number, Normal Reynolds Number and the angle of inclination of the jets, quantitatively dictates the transition.

[1]Presently at Department of Physics, Danish Technical University, Denmark

9:05AM M30.00006 Hydrodynamic Mass of Bluff Bodies with a Cavity[1] , MOHAMED ELGABAILI, California State University, Northridge, KENNETH DESABRAIS, US Army NSRDEC, HAMID JOHARI, California State University, Northridge — Hydrodynamic mass of an object may be used to compute the contribution of unsteady drag resulting from potential flow. Even though the hydrodynamic mass of certain bluff bodies such as cylinder and sphere have been available from analytical considerations for a long time, there are no analytical solutions for a general bluff body with a cavity such as a cup facing the flow or a round parachute canopy. There is, however, an analytical solution for spherical shells of various concavities. The translational hydrodynamic mass of cups having various depth and thickness as well as round parachute canopies during inflation was computed using a finite element solver. The kinetic energy of the potential flow around the body was used to extract the hydrodynamic mass. Results indicate that the hydrodynamic mass of a cup can be decomposed into two components, the hydrodynamic mass of a cylinder whose axis is aligned with the flow and the mass of fluid within the cup cavity. Similarly, the hydrodynamic mass of a parachute canopy during various stages of inflation may be written as the hydrodynamic mass of a disk having the same area as the projected area of the canopy plus the mass of fluid enclosed by the canopy.

[1]Sponsored by the US Army Natick RDEC.

9:18AM M30.00007 Explosion cavities , ADRIEN BENUSIGLIO, CHRISTOPHE CLANET, LadHyX - Ecole Polytechnique, DAVID QUERE, PMMH - ESPCI — We study the cavities produced at the water-air interface by the explosion of firecrackers. Without confinement, we first observe a spherical hole which grows and reaches a maximal size that depends on the initial energy. Beyond this maximal extension, the cavity collapses in an anisotropic way and leads to the formation of a jet right at the point of explosion. In the case of a confined explosion in a cylindrical tube, the water/air interface initially moves away from the explosion location and reaches a maximal depth that again depends on the initial energy of the explosion. The main difference with the unconfined limit is the fact that the size of the cavity cannot be larger than the size of the confining tube.

9:31AM M30.00008 Rarefied gas correction for the bubble entrapment singularity in drop impacts , LAURENT DUCHEMIN, IRPHE, CHRISTOPHE JOSSERAND, Institut d Alembert — We study the non-continuous correction in the dynamics of drop impact on a solid substrate. Close to impact, a thin film of gas is formed beneath the drop so that the local Knudsen number is of order one. We consider the first correction to the dynamics which consists of allowing slip of the gas along the substrate and the interface. We focus on the singular dynamics of entrapment that can be seen when surface tension and liquid viscosity can be neglected. There we show that different dynamical regimes are present that tend to lower the singularity strength. We finally suggest how these effects might be connected to the influence of the gas pressure in the impact dynamics observed in recent experiments.

9:44AM M30.00009 The meandering instability of a partial wetting rivulet , STÉPHANIE COUVREUR, ADRIAN DAERR, Paris Diderot University — When a liquid rivulet flows down an inclined plate in partial wetting conditions, it can takes different kinds of shapes. For small flow speed rates, the rivulet flows down the gravity direction, straight along the steepest slope. When increasing the flow rate, an instability leads to the growth of curves and the rivulet finally adopts a sinuous stationary shape, we call this the meandering regime. The rivulet bends grow from defects of the contact line: when the liquid flows along a small perturbation, it is submitted to an inertial centrifugal force which tends to move it to the exterior side of the small curve. Below the instability threshold this force merely distorts the rivulet cross section away from its symetric circular shape at rest, but the contact line remains pinned.When increasing the speed of the liquid (by increasing the flow rate), the inertial effect becomes higher and higher, increasing the external contact angle, until it reaches its critical value, whereupon the rivulet moves and the instability grows. We will show experiments that the critical flow rate depends strongly on the geometry of the initial rivulet. We will explain this dependency through a force balance model, in quantitative agreement with our experiments. S. Couvreur and A. Daerr, EPL, 2012

9:57AM M30.00010 Sparsity-promoting Dynamic Mode Decomposition[1] , MIHAILO JOVANOVIC, University of Minnesota, PETER SCHMID, LadHyX, CNRS/Ecole Polytechnique, France — Dynamic mode decomposition (DMD) represents an effective means for capturing the essential features of numerically or experimentally generated flow fields. In order to strike a balance between the quality of approximation (in the least-squares sense) and the number of modes that are used to approximate the given fields, we develop a sparsity-promoting version of the standard DMD algorithm. This is achieved by combining tools and ideas from convex optimization and the emerging area of compressive sensing. Several examples of flow fields resulting from simulations and experiments are used to illustrate the effectiveness of the developed method.

[1]This work was performed during the 2012 Summer Program at the Center for Turbulence Research with financial support from Stanford University and NASA Ames Research Center.

Tuesday, November 20, 2012 8:00AM - 10:10AM –

Session M31 Wind Energy II 33B - Chair: Raul Cal, Portland State University

8:00AM M31.00001 Spatial characterization of the turbulent structure of a model wind turbine: high speed PIV measurements , JIAN SHENG, Texas Tech Univ., LEONARDO CHAMORRO, SEUNG-JAE LEE, ROGER ARNDT, FOTIS SOTIROPOULOS, U. of Minneasota — Wind turbine wakes are complex flow structures that are modulated by the wind turbines and the characteristics of the approach flow, among others. Determining and quantifying the dominating processes that modulate their behavior is essential to improve wind farm design. High speed Particle Image Velocimetry was used to characterize the temporal and spatial features of a model wind turbine wake in their axis of symmetry in the near and far wake field. The model turbine was placed in a boundary layer flow developed in a wind tunnel under neutrally stratified conditions. The high speed measurements allowed us to determine the evolution of a range of coherent structures and their interactions with the surrounding flow. Multi-correlations and spectra as well as the spatial distribution of turbulence quantities provide relevant information on the key turbulent mechanisms that modulate a turbine wake and regulate turbulent transport.

8:13AM M31.00002 Turbulence effects on a full-scale 2.5 MW horizontal axis wind turbine[1] , LEONARDO CHAMORRO, SEUNG-JAE LEE, DAVID OLSEN, CHRIS MILLIREN, JEFF MARR, ROGER ARNDT, FOTIS SOTIROPOULOS, University of Minnesota — Power fluctuations and fatigue loads are among the most significant problems that wind turbines face throughout their lifetime. Turbulence is the common driving mechanism that triggers instabilities on these quantities. We investigate the complex response of a full-scale 2.5 MW wind turbine under nearly neutral thermal stratification. The study is performed in the EOLOS Wind Energy Research Field Station of the University of Minnesota. An instrumented 130 meter meteorological tower located upstream of a Clipper Liberty C96 wind turbine is used to characterize the turbulent flow and atmospheric conditions right upstream of the wind turbine. High resolution and synchronous measurements of the wind velocity, turbine power and strain at the tower foundation are used to determine the scale-to-scale interaction between flow and the wind turbine. The structure of the fluctuating turbine power and instantaneous stresses are studied in detail. Important insights about the role of turbulent and coherent motions as well as strong intermittent gusts will be discussed.

[1]Funding was provided by Department of Energy DOE (DE-EE0002980) and Xcel Energy through the Renewable Development Fund (grant RD3-42).

8:26AM M31.00003 A wind tunnel study on the effects of complex topography on wind turbine performance , KEVIN HOWARD, St. Anthony Falls Laboratory, Dep. Civil Engineering, UMN, STEPHEN HU, Aerospace Engineering and Mechanics, UMN, LEONARDO CHAMORRO, MICHELE GUALA, St. Anthony Falls Laboratory, Dep. Civil Engineering, UMN — A set of wind tunnel experiments were conducted to study the response of a wind turbine under flow conditions typically observed at the wind farm scale, in complex terrain. A scale model wind turbine was placed in a fully developed turbulent boundary layer flow obtained in the SAFL Wind Tunnel. Experiments focused on the performance of a turbine model, under the effects induced by a second upwind turbine or a by three-dimensional, sinusoidal hill, peaking at the turbine hub height. High frequency measurements of fluctuating streamwise and wall normal velocities were obtained with a X-wire anemometer simultaneously with the rotor angular velocity and the turbine(s) voltage output. Velocity measurements in the wake of the first turbine and of the hill were used to determine the inflow conditions for the downwind test turbine. Turbine performance was inferred by the mean and fluctuating voltage statistics. Specific experiments were devoted to relate the mean voltage to the mean hub velocity, and the fluctuating voltage to the unsteadiness in the rotor kinematics induced by the perturbed (hill or turbine) or unperturbed (boundary layer) large scales of the incoming turbulent flow. Results show that the voltage signal can be used to assess turbine performance in complex flows.

8:39AM M31.00004 PIV and Acoustic Investigation for a 2D Wind Turbine Airfoil[1] , GUANNAN WANG, MARK GLAUSER, Syracuse University — This study investigated the aerodynamic characteristics of a 2D airfoil designed for wind turbine applications using PIV and surface pressure measurements. The experiments were carried out in a low speed wind tunnel with/without large scale unsteadiness in the flow and with/without active closed loop blowing control on the suction surface of the airfoil. This study also measured the acoustic signal emitted from the same type of airfoil with six far field microphones in an anechoic chamber and the results indicated that the unsteadiness in the freestream affected the noise characteristics of the airfoil significantly.

[1]Funded by DoE through University of Minnesota Wind Energy Consortium.

8:52AM M31.00005 Efficiency and flow structure of vertical-axis turbines with an upstream deflecting plate[1] , DAEGYOUM KIM, MORTEZA GHARIB, California Institute of Technology — The power generation and flow structure of straight-bladed vertical-axis turbines with an upstream deflector are investigated experimentally in tunnel facilities. When an upstream deflecting plate is normal to flow direction, a region of low velocity is formed in its near-wake. However, the flow speed outside the near-wake region becomes higher than the free-stream speed. Since blades outside the wake encounter higher flow velocity, they can rotate with higher torque and rotating speed compared to the case without an upstream deflector, which results in power output increase. Here, we study the effect of deflector position and width on the efficiency of vertical turbines. We also discuss the flow structure generated by the deflector system.

[1]This research is supported by the Gordon and Betty Moore foundation.

9:05AM M31.00006 Start-up dynamics of vertical axis turbines[1] , KATHERINE TAYLOR, JOHN DABIRI, Caltech — We present an experimental study of the self-starting behavior of vertical axis turbines, in order to guide the design of systems that operate in unsteady flows. The torque, angular velocity, and power generation of a scale model turbine were measured in a free surface water tunnel for different starting angles of the rotor blades and for different flow speeds. The starting behavior of the turbine was found to be sensitively dependent on the initial angle of the rotor at low flow speeds. A conceptual model was developed in order to explain the observed behavior in terms of the instantaneous lift and drag on the rotor blades.

[1]Funding provided by the Gordon and Betty Moore Foundation

9:18AM M31.00007 The Influence of Rotor Configurations on the Energy Production in an Array of Vertical-Axis Wind Turbines[1] , MATTHIAS KINZEL, DANIEL ARAYA, JOHN DABIRI, Caltech — We analyze the flow field within an array of 18 vertical-axis wind turbines (VAWTs) at full-scale and under natural wind conditions. The emphasis is on the energy flux into the turbine array and the energy extraction by the turbines. 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 a detailed insight into the turbine wakes and the recovery of the flow. A high planform kinetic energy flux is detected, which enables the flow velocities to return to 95% of the upwind value within six rotor diameters downwind from a turbine row. This is significantly faster than the recovery behind a typical horizontal-axis wind turbine (HAWT). The Presentation will compare the results for different rotor configurations. Conclusions will be drawn about the influence of these configurations on the power production of the individual turbines as well as the turbine array as a whole.

[1]The authors gratefully acknowledge funding from the National Science Foundation Energy for Sustainability program (Grant No. CBET-0725164) and the Gordon and Betty Moore Foundation.

9:31AM M31.00008 Effect of turbulence intensity on power generation in a 4x3 wind turbine array[1] , MURAT TUTKUN, Norwegian Defense Research Establishment, ELIZABETH CAMP, RAUL BAYOAN CAL, Portland State University, Department of Mechanical and Materials Engineering — Turbulence intensity is highly variable in the atmospheric boundary layer. This is the typical environment where wind farms are placed and operate. The characteristics of the turbulence have the ability to impact the power production of wind turbines. Here, a wind tunnel study on a 4x3 wind turbine array is performed in which the turbulence intensity is varied independently. These experiments are carried out in a wind tunnel setting and the power is measured using torque and angular frequency sensing devices. The levels of turbulence are varied via a dynamic grid. Three cases are obtained one passive and two active (with two distinct ranges). Power production along the centerline of the array was measured for each of the cases using model turbines outfitted with torque sensors.

[1]This work is funded by the National Science Foundation grants CBET-1034581 and ECCS-1032647 as well as Institute for Sustainable Solutions at Portland State University.

9:44AM M31.00009 Direct power measurements on wind turbine array configurations , DOMINIC DELUCIA, RAUL BAYOAN CAL, Portland State University — The reliability on the power extraction through wind turbines is an area of need given the increasing size of the arrays and energy demand. The turbulence effects generated by wind turbines on the subsequent rows downstream are assessed. Mechanical torque on the hubs of the model wind turbine is recorded and the power is calculated, where the measurements are performed in the Portland State University wind tunnel. Simultaneous torque and angular frequency of the rotors is record at three locations in a 3 by 4 wind turbine array. In this study, the effects due to in-line and staggered configurations are investigated. The base case configuration is a 3 by 4 array with a 6D downstream spacing and a 3D transverse spacing. The results are compared to wind turbine arrays of different spacing configurations. The trends in the data suggest the power is significantly increased when the downstream position are offset by 1.5D in the transverse direction not only for subsequent turbines but also when the turbines are staggered.

9:57AM M31.00010 On the characteristic features of wind-turbine tip vortices: A wind tunnel experiment , DAVID GREEN, LEONARDO CHAMORRO, ROGER ARNDT, FOTIS SOTIROPOULOS, U. of Minnesota, JIAN SHENG, Texas Tech Univ. — Understanding the complex interaction between the vortical flow structures shed by Horizontal Axis Wind Turbines (HAWT) and the turbulent flow is crucial to optimize blade design and momentum recovery in the turbine wake, which defines the wind farm layout. Tip vortices shed by the blades play a key role in shaping up the wake behind a HAWT. Phase-locked Particle Image Velocimetry (PIV) is employed to measure mean wake, flow fluctuations, and subsequently identify large-scale coherent flow structures. Twelve consecutive downstream locations up to twelve rotor diameters and three upstream locations up to three rotor diameters are measured. Experiments are conducted at Reynolds numbers of Re=3x10^5, 4x10^5 and 12x10^5 based on the rotor diameter. To achieve sufficient spatial resolution, two fields are taken at each streamwise location to cover the upper and lower half of the model turbine. It is found that tip vortices above the turbine hub have clear structural identity as they are advected downstream. Instead of following the expanding trend of mean wake, they converge. In the lower region, the identities of tip vortices remain only one rotor diameter downstream and merge into the mean near wake. Robust statistical analysis on velocity fluctuations, Reynolds stresses, and TKE budget will be discussed.

Tuesday, November 20, 2012 8:00AM - 10:10AM –

Session M32 Granular Flows IV 33C - Chair: Elisabeth Guazzelli, Aix Marseille University

8:00AM M32.00001 What is the granular response to a high-speed impact?[1] , ABE CLARK, LOU KONDIC, R.P. BEHRINGER, Duke University — Although many studies of impact on a granular material exist, the connections between the local granular response, the microscopic processes which dissipate kinetic energy, and the intruder dynamics are unclear, largely due to experimental difficulties in obtaining very fast data at the grain scale. We use high-speed imaging (40 kHz) of an intruder striking a quasi-2D system of photoelastic disks, yielding both the intruder dynamics and the force response of individual grains. The frame rates are fast enough to resolve rich acoustic activity on the particle scale. For long time scales, the intruder dynamics are consistent with previously used empirical force laws. However, for short time scales, we observe very large fluctuations in the deceleration, which we connect to the intermittent acoustic activity beneath the intruder as it moves. We show that these intense, intermittent acoustic pulses, which travel much faster than the intruder along networks of grains, are the primary microscopic mechanism of energy loss. These pulses carry energy away into the medium, and they decay roughly exponentially with distance. We examine the statistics of these fluctuations in order to better understand their origin and behavior.

[1]Supported by the US DTRA under grant HDTRA1-10-0021.

8:13AM M32.00002 Particles impacting on a granular bed , JOHN HINCH, DAMTP, University of Cambridge — An asymptotic analysis is made to find the penetration depth and the stopping time for a particle impacting a granular bed. Newton's equation is solved with a drag force with two terms, one term proportional to the square of the velocity and one term linear in the depth. The penetration depth is found to increase with the logarithm of the impact velocity, while the stopping time is found to decrease with the inverse of the square root of the logarithm of the impact velocity.

8:26AM M32.00003 Flow-mediated coupling on projectiles falling in a superlight granular medium , GABRIEL A. CABALLERO-ROBLEDO, CINVESTAV-Monterrey, Nuevo Leon, Mexico, JUAN M. SOLANO-ALTAMIRANO, Department of Chemistry, University of Guelph, Ontario, Canada, VINCENT KAMPHORST, Physics of Fluids Group, University of Twente, Enschede, The Netherlands, FELIPE PACHECO-VÁZQUEZ, GRASP, Physics Department, University of Liège, Belgium, J.C. RUIZ-SUÁREZ, CINVESTAV-Monterrey, Nuevo Leon, Mexico — Interesting collective motion emerges when several heavy disks fall in a quasi 2D granular bed of extremely light grains [F. Pacheco-Vázquez and J.C. Ruiz-Suárez, *Nat. Comms.* **1**, 123 (2010)]. In particular, when two disks impact side by side they initially repel, then they attract each other, until they finally stop. We perform experiments and Discrete Element Soft-Particle simulations to determine the range of action and the origin of these attractive and repulsive flow-mediated forces. Our findings suggest that repulsion results from jamming of grains between intruders while attraction would be due to a "granular pressure" drop in the region between intruders caused by a high flow velocity of grains: a Bernoulli-like effect.

8:39AM M32.00004 Force measurements after granular impact using instrumented spheres , SYLVAIN JOUBAUD, Laboratoire de Physique - ENS de Lyon, TESS HOMAN, Physics of Fluids, University of Twente, YOANN GASTEUIL, Laboratoire de Physique - ENS de Lyon, DETLEF LOHSE, DEVARAJ VAN DER MEER, Physics of Fluids, University of Twente — Impacts of solid spheres on soft and dry sand may result in the rapid sinking of the sphere into the sand. This in turn can lead to a jet shooting up from the surface of the sand. The dynamics of the sphere is affected by the impact velocity and the ambient air pressure. In this work, we performed direct measurement of the acceleration using instrumented spheres from *Smart*$\mathbf{INST}$. These spheres non-invasively and continuously transmit the force acting on themselves as they penetrate into the sand using a radio signal, *i.e.*, without disturbing the behavior of the granular medium which any invasive measurement technique would do. The signal reveals very complex dynamics including a signature of the jet formation. These results are compared to various phenomenological models.

8:52AM M32.00005 Fill-level symmetry and minimization of energy states in rotating tumblers with polygonal cross-sections[1] , NICHOLAS A. POHLMAN, DANIEL F. PAPROCKI, JR., YUN SI, Northern Illinois University — Typically in rotating tumblers, constant rotation rates and circular cross-sections are used as they jointly produce a steady, uniform flowing layer at the free surface. On the other hand, experiments conducted in polygon-shaped tumblers produce unsteady conditions due to the rapidly changing flowing layer length. Results analyzing free surface properties indicate that the particle dynamics within the flowing layer attempt to minimize energy of the flowing system: The arithmetic difference between the angle of repose and the tumbler orientation has a functional relationship with the instantaneous flowing layer length in the form of a catenary. The peaks of the catenary are affected by the number of sides on the polygon cross-section as well as the symmetry around the critical 50% fill fraction. Furthermore, oscillation of the flowing layer position appears to affect the free surface curvature. This result is likely due to the rapidly increasing and decreasing length of the free surface and the rotational inertia of particles entering the flowing layer.

[1]Funding provided by NIU's Office of Student Engagement and Experiential Learning

9:05AM M32.00006 Slow axial drift in three dimensional tumblers , ZAFIR ZAMAN, Northwestern University, UMBERTO D'ORTONA, Universite d Aix-Marseille, PAUL UMBANHOWAR, JULIO M. OTTINO, RICHARD M. LUEPTOW, Northwestern University — We recently demonstrated the existence of coherent axial drift of monodisperse particles in partially filled spherical tumblers from DEM simulations and experiments. This motion occurs solely in the flowing layer: particles move gradually toward the pole at the top of the flowing layer and toward the equator at the bottom of the flowing layer. The drift is small relative to streamwise displacements, thus particles require many passes through the flowing layer to progress from the equator to the pole and back. Since axial drift is negligible in cylindrical containers except near the endwalls, this suggests that axial variation in tumbler diameter is required for axial drift. To understand how axial variations in flowing layer length, L, determines drift, we conducted new experiments and simulations of partially filled double cone tumblers of varying wall slope. Axial drift remains present in the conical geometry, and the drift speed increases with the equator diameter for fixed tumbler length. Results from both the double cone and the spherical tumbler reveal that the axial drift velocity depends on L and the axial position. Funded by NSF Grant CMMI-1000469.

9:18AM M32.00007 Competing segregating effects of gravity and shear rate gradients in dense granular flows in a drum: theory and simulations , KIMBERLY HILL, DANIELLE TAN, University of Minnesota — A well-known rule of thumb for sheared mixtures of different-sized (same density) particles is that 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." However, it has been recently shown that in a sheared granular mixture, larger particles may rise or fall relative to the small particles, or even rise only partway to some steady-state height in a sheared mixture. We present a theory that accounts for this complex behavior as a balance between gravity-driven segregation effects and, effectively, granular temperature gradients driven by the shear. Then, we test this theory using discrete element method simulations of different mixtures rotated in a partially-filled drum. Using the theory and simulations, we show that for all mixtures we test, the segregation fluxes are driven by the difference between the partitioning of kinetic and contact stresses among the species in the mixture subjected to a gravity-induced contact stress gradient. Specifically, all particles bear a fraction of the local contact stress equal to their local concentration in the mixture, but the smaller particles bear a higher fraction of the local kinetic stress (akin to granular temperature). This presents a new physical mechanism for kinetic sieving: even where the total granular temperature is small, the higher granular temperature of the smaller particles segregates them downward, in the direction of gravity, towards high stress and low temperature regions.

9:31AM M32.00008 Investigation of the mobile granular layer in bed-load transport[1] , ELISABETH GUAZZELLI, PASCALE AUSSILLOUS, Aix Marseille Univ., CNRS, IUSTI UMR 7343, JULIEN CHAUCHAT, LEGI, UJF/INPG/CNRS, MICKAEL PAILHA, LOCIE, CNRS - Univ. de Savoie, MARC MEDALE, Aix Marseille Univ., CNRS, IUSTI UMR 7343, AIX MARSEILLE UNIVERSITÉ, CNRS, IUSTI UMR 7343 TEAM, LEGI, UJF/INPG/CNRS TEAM, LOCIE, CNRS - UNIVERSITÉ DE SAVOIE TEAM — The mobile layer of a granular bed composed of spherical particles is experimentally investigated in a laminar rectangular-channel flow. Both particle and fluid velocity profiles are obtained using particle image velocimetry for different index-matched combinations of particles and fluid. While the Shields number controls incipient motion, it is not the most appropriate parameter for describing bed-load transport. The experimental observations suggest that the appropriate length-scale is the fluid height and that the proper control parameter is the dimensionless fluid flow-rate. A two-phase continuum model having a frictional rheology to describe particle-particle interactions can capture most of the experimental observations. Rheological constitutive laws having increasing degree of sophistication are discussed.

[1]Funding from Agence Nationale de la Recherche (Project Dunes ANR-07-3 18-3892) is gratefully acknowledged.

9:44AM M32.00009 Shear-Induced Diffusion in a Dense Frictional Disk Packing , JOSHUA DIJKSMAN, JIE REN, ROBERT BEHRINGER, Duke University — We study shear-induced diffusion in a dense disordered packings of frictional photoelastic disks. We induce diffusion by subjecting the packing to uniform oscillatory shear cycles. We can track both displacement and rotational motion, and measure interparticle forces obtained from the photoelastic response of the disks. The shear deformation is volume conserving, so each experiment corresponds to a well defined density. We then vary the density to probe its impact on diffusion; we also study the influence of the shear amplitude. Surprisingly, we find that both rotational and translational diffusion increases with density for all but the highest densities – clearly steric hindrance only becomes relevant at the highest packing fractions. At the onset of mechanical stability, as indicated by an increase in the total pressure in the system, we also find profound changes in both the amplitude, time dependence and directionality of the particle diffusion.

9:57AM M32.00010 Compression and shear in ultra low friction soft sphere packings: an experimental exploration[1] , HU ZHENG, Tongji University, JOSHUA DIJKSMAN, ROBERT BEHRINGER, Duke University — We study the quasi-static deformation of three dimensional sphere packings close to the onset of mechanical rigidity. We perform the experiments on slightly polydisperse, nearly frictionless soft hydrogel spheres in a tri-axial shear apparatus. We can access both system pressure and structural flow information by index matched scanning on the submersed, transparent hydrogel spheres. 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 com pressed packings.

[1]Acknowledgements: This work supported by NSF grant DMR12-06351, and ARO grant W911NF-11-1-0110.

Tuesday, November 20, 2012 10:30AM - 11:05AM –

Session N33 Invited Session: Waves and Wave-driven Flow on a Coral Reef Ballroom 20A - Chair: Kraig Winters, University of California, San Diego

10:30AM N33.00001 Waves and wave-driven flow on a coral reef[1] , STEPHEN MONISMITH, Stanford University — It has been long appreciated that surface wave breaking is a primary mechanism for driving flows over coral reefs and so influences a wide variety of reef ecological processes. In this talk I will discuss measurements of waves and wave-driven flows made on the north shore of Moorea, FP. Despite the steep slope and large wave steepness, integral properties of the waves we observe match linear longwave theory to a remarkable extent, although their vertical structure does seem to differ from what is expected from theory. Our observations also show that the net transport over the reef is carried by both Stokes drift and a mean Eulerian flow, although the portioning changes as the waves shoal, break and dissipate. The balance between mean setup due to breaking, which also matches simple theory, and friction inshore of the surfzone/reef crest sets the overall flow rate. While simple theories match the observations quite well, their predictive value is somewhat reduced by the fact that they include 3 parameters that must be found empirically because they involve the basic geometry of the reef and the complex nature of frictional resistance associated with reef roughness.

[1]0622967 for their support.

Tuesday, November 20, 2012 10:30AM - 11:05AM –

Session N34 Invited Session: Simulation-based Planning of Surgical Interventions in Pediatric Cardiology Ballroom 20BC - Chair: Stefan Llewellyn Smith, University of California, San Diego

10:30AM N34.00001 Simulation-based planning of surgical interventions in pediatric cardiology , ALISON MARSDEN, University of California San Diego — Hemodynamics plays an essential role in the progression and treatment of cardiovascular disease. This is particularly true in pediatric cardiology, due to the wide variation in anatomy observed in congenital heart disease patients. While medical imaging provides increasingly detailed anatomical information, clinicians currently have limited knowledge of important fluid mechanical parameters. Treatment decisions are therefore often made using anatomical information alone, despite the known links between fluid mechanics and disease progression. Patient-specific simulations now offer the means to provide this missing information, and, more importantly, to perform in-silico testing of new surgical designs at no risk to the patient. In this talk, we will outline the current state of the art in methods for cardiovascular blood flow simulation and virtual surgery. We will then present new methodology for coupling optimization with simulation and uncertainty quantification to customize treatments for individual patients. Finally, we will present examples in pediatric cardiology that illustrate the potential impact of these tools in the clinical setting.

Tuesday, November 20, 2012 11:10AM - 11:30AM –

Session P33 Andreas Acrivos Dissertation Award Lecture Ballroom 20A - Chair: Shelley Anna, Carnegie Mellon University

11:10AM P33.00001 Andreas Acrivos Dissertation Prize Lecture: Phytoplankton in Flow , WILLIAM M. DURHAM, Department of Zoology, University of Oxford — Phytoplankton are small, unicellular organisms that form the base of the marine food web and are cumulatively responsible for half the global oxygen production. While phytoplankton live in an environment characterized by ubiquitous fluid flow, the impact of hydrodynamic conditions on their ecology remain poorly understood. In this talk, I report on two novel biophysical mechanisms based on the interaction between phytoplankton motility and fluid shear. First, I will consider "thin phytoplankton layers," important hotspots of ecological activity that are found meters beneath the ocean surface and contain cell concentrations up to two orders of magnitude above ambient. Using a combination of experiments, individual-based simulations, and continuum modeling, we have shown that layers can form when the vertical migration of phytoplankton is disrupted by hydrodynamic shear. This mechanism - which we call "gyrotactic trapping" - is capable of triggering thin phytoplankton layers under hydrodynamic conditions typical of the environments that often harbor thin layers. Second, I will discuss the potential for turbulent shear to produce patchiness in the spatial distribution of motile phytoplankton. Field measurements have revealed that motile phytoplankton form aggregations at the Kolmogorov scale, whereas non-motile cells do not. We propose a new mechanism for the formation of this small-scale patchiness based on the interplay of gyrotactic motility and turbulent shear. Using laboratory experiments, an analytical model of vortical flow, and isotropic turbulence generated via Direct Numerical Simulations, we found that motile phytoplankton rapidly aggregate, whereas non-motile cells remain randomly distributed. Taken together, these two mechanisms demonstrate that the interaction of cell motility with flow plays a fundamental role in phytoplankton ecology and, as a consequence, can contribute to shape macroscale characteristics of the ocean.

Tuesday, November 20, 2012 11:10AM - 11:30AM –

Session P34 Francois Frenkiel Award Lecture Ballroom 20BC - Chair: P.K. Yeung, Georgia Tech

11:10AM P34.00001 Francois Frenkiel Award Lecture: Folded micro-threads: Role of viscosity and interfacial tension[1] , THOMAS CUBAUD, Stony Brook University — Viscous threads belong to a class of flow structures having common characteristics between miscible and immiscible fluid streams. Here, we examine the interplay between flows and microgeometries for passively destabilizing high-viscosity fluid threads. The shape and evolution of periodically folded threads are experimentally investigated in a microfluidic network. The fluidic system is designed for the production and lubricated transport of very uniform folds. To study the influence of viscosity and interfacial tension on buckling deformations, multiphase flows are scrutinized using both miscible and immiscible fluid pairs. The parameters used to analyze folding morphologies include thread diameter, arc length, fold amplitude, and wavelength. When fluids are immiscible, the onset of viscous folding is characterized as a function of the capillary number and the phenomenon of capillary unfolding where a corrugated thread straightens along the flow direction is demonstrated. The spatial transition from folding to coiling-like flow behavior of highly viscous capillary threads is also shown.

In collaboration with Bibin M. Jose and Samira Darvishi, Stony Brook University.

[1]This work is supported by NSF (CBET-0932925).

Tuesday, November 20, 2012 1:00PM - 2:57PM –

Session R1 Geophysical: Ocean IV 22 - Chair: Chris Rehman, Iowa State University

1:00PM R1.00001 The effects of a shear flow on lee waves , MICHAEL PATTERSON, University of Bath, STUART DALZIEL, DAMTP, University of Cambridge, COLM CAULFIELD, BP Institute & DAMTP, University of Cambridge, STÉPHANE LE BRUN, DAMTP, University of Cambridge & École Polytechnique, France — We explore experimentally and theoretically the effects of a shear flow on lee waves that are generated by a stationary isolated three-dimensional obstacle in a low-Froude-number stratified flow. We observe that the uniform flow (beneath the shear layer) can be divided into two regions: an essentially two-dimensional flow around the base of the obstacle and a wave-generating flow over the top portion of the obstacle. The third region's structure is dependent on the sign of the shear, and is either a wave-free region above the critical height where the wave speed equals the local fluid speed or a region in which the waves are fully reflected. By separating the permanent waves produced into distinct categories, we compare detailed experimental measurements with theoretical predictions. Finally we examine the time-dependent establishment of the waves, including the transition from a uniform flow to a sheared flow.

1:13PM R1.00002 Spatial structure of tidally generated internal waves[1] , MATTHEW PAOLETTI, AMADEUS DETTNER, MATTHEW DRAKE, HARRY L. SWINNEY, University of Texas at Austin — Tidal flow over bottom topography is one of the main sources of internal wave energy in the ocean, which may be converted into gravitational potential energy through mixing when the internal waves break. Internal wave breaking can occur when the destabilizing vertical shear of the waves overcomes the stabilizing effects of gravity. While past studies have determined the conversion rate of the tidal motions into internal wave energy, a general understanding of the spatial structure and shear profiles of tidally generated internal waves is lacking. Here, we present 2D experimental and computational studies of internal wave generation by tidal flow over several types of topographic ridges. For each topographic profile, we vary the criticality parameter, which is the ratio of the topographic slope to the wave beam slope, by independently changing the tidal frequency, stratification, and topographic slope. We also consider cases where the topography is beneath a turning depth, below which internal waves are evanescent owing to the weak stratification. The spatial structure of the internal waves is characterized by the velocity amplitude, principal wavenumber, width, and the local Richardson number, which determines the stability properties.

[1]Supported by ONR MURI Grant N000141110701.

1:26PM R1.00003 Energy flux of internal waves generated by tidal flow over topography beneath a turning depth[1] , MATTHEW DRAKE, M.S. PAOLETTI, F.M. LEE, P.J. MORRISON, H.L. SWINNEY, University of Texas Austin — We present experimental and computational studies of internal gravity wave generation by tidal flow over 2D topography in a stably stratified fluid designed to model the deep ocean. King et al. found that there exist regions in the deep ocean where the buoyancy frequency (proportional to the square root of the density gradient) becomes less than the tidal frequency [King et al., J. Geophys. Res. 117, C04008 (2012)]. Below such "turning depths" the internal gravity waves become evanescent. The effect of turning depths on global internal wave generation has not been examined. Here we present experiments and 2D Navier-Stokes simulations that determine the far-field energy flux as a function of the distance of the turning depth above the topography. We examine how the energy flux depends on the tidal frequency, stratification, topographic profile, and the distance of the topography below the turning depth.

[1]Supported by ONR MURI Grant N000141110701.

1:39PM R1.00004 Generation of internal waves and boundary currents by tidal flow over 2D topography[1] , AMADEUS DETTNER, MATTHEW PAOLETTI, HARRY L. SWINNEY, University of Texas at Austin — The majority of internal wave energy in the ocean is produced by tidal flow over topography. Regions of critical topography, where the topographic slope is equal to the slope of the internal waves, is often believed to contribute most significantly to the radiated internal wave power. Here, we present 2D experimental and computational studies of internal wave generation by tidal flow over several types of topographic ridges. We vary the criticality parameter ϵ, which is the ratio of the topographic slope to the wave beam slope, by independently changing the tidal frequency, stratification and topographic slope, which allows us to study subcritical ($\epsilon < 1$), critical ($\epsilon = 1$), and supercritical topography ($\epsilon > 1$). As in prior work [Zhang et al., *Phys. Rev. Lett.* (2008)], we observe resonant boundary currents for $\epsilon = 1$. However, we find that the normalized radiated power monotonically increases with ϵ. We find that an appropriate normalization condition leads to a universal scaling of the radiated power as a function of ϵ.

[1]Supported by ONR MURI Grant N000141110701.

1:52PM R1.00005 Tidal conversion by a periodic array of ridges[1] , LIKUN ZHANG, MATTHEW PAOLETTI, HARRY SWINNEY, University of Texas at Austin — The generation of internal waves by tidal flow over submarine topography such as ridges is the main source of internal tidal energy in the oceans. For multiple ridges the dependence of the radiated power on the ridge height and slope is different from that for an individual ridge, and for multiple ridges the power also depends on the spacing between the topographic features. Prior numerical and analytical studies on tidal generation by a periodic array of steep ridges suggest the possible saturation of conversion rate when the dimensionless height of the topography is large [S. Khatiwala, Deep-Sea Res. (2003); J. Nycander, J. Fluid. Mech. (2006); N. J. Balmforth and T. Peacock, J. Phys. Ocean. (2009)]. Here we perform 2D Navier-Stokes simulations and laboratory experiments on the generation of internal tides from a periodic array of supercritical barriers. We determine how the radiated power and saturation depend on the tidal flow, topography, and stratification.

[1]Supported by ONR MURI Grant N000141110701

2:05PM R1.00006 Calculating viscous internal gravity waves , STEFAN LLEWELLYN SMITH, MAE, UCSD — Internal gravity waves (IGWs) are ubiquitous features of the ocean and atmosphere, thought to be critical in global energy budgets. There has hence been much interest in developing simple models of IGW generation. However, only a few special solutions for simple geometries are known, and the hyperbolic spatial nature of the governing equations leads to numerical difficulties. At the same time, modern developments in laboratory techniques now reveal the quantitative effects of viscosity in experiments on IGW generation by oscillating bodies. Even fewer solutions are known when viscosity is present. We discuss boundary integral methods for viscous IGW generation, and present some results and applications.

2:18PM R1.00007 Forcing of oceanic mean flows by dissipating internal tides[1] , NICOLAS GRISOUARD, OLIVER BUHLER, Courant Institute of Mathematical Sciences - New York University — We present a theoretical study of the effective mean force exerted on an oceanic mean flow due to the presence of small-amplitude internal waves that are forced by a barotropic tide flowing over a topography and are also subject to dissipation. Although the details of our computation are quite different, we recover the main action-at-a-distance result familiar from atmospheric wave-mean interaction theory, namely that the effective mean force that is felt by the mean flow is located in regions of wave dissipation, and not necessarily near the topographic wave source. Specifically, using a perturbation series in small wave amplitude, we compute the three-dimensional leading-order wave field using a Green's function approach, derive an explicit expression for the leading-order effective mean force at the next order within the framework of generalized Lagrangian-mean theory, discuss in detail the range of situations in which a strong, secularly growing mean-flow response can be expected, and finally compute the effective mean force numerically in a number of illustrative examples with simple topographies.

[1]Financial support under the United States National Science Foundation grants NSF/OCE 1024180 and NSF/DMS 1009213 is gratefully acknowledged.

2:31PM R1.00008 Effect of slope criticality and tidal forcing on internal tide energetics at a model ridge , NARSIMHA RAPAKA, BISHAKHDATTA GAYEN, SUTANU SARKAR, University of California San Diego — Direct and large eddy simulations are performed to study the internal waves generated by the oscillation of a barotropic tide over a model ridge of triangular shape. The criticality parameter, defined as the ratio of the topographic slope to the characteristic slope of the tidal rays, is varied from subcritical to supercritical values. The barotropic tidal forcing is also systematically increased. Higher baroclinic modes are generated with increasing criticality parameter, resulting in generation of intensified beams near the topography in critical and supercritical cases. The radiated internal wave energy flux increases from subcritical to supercritical cases in laminar flow regime. In critical and supercritical cases with higher forcing, there is turbulence and significant reduction (as much as 25%) of the radiated wave flux with respect to laminar flow results. Analysis of the baroclinic energy budget shows that the decrease in the radiated wave flux is associated with a decrease in energy conversion from the barotropic to baroclinic flow, caused by increased drag and mixing of momentum near the ridge, and additionally because of conversion to turbulence.

2:44PM R1.00009 Numerical Simulation of Internal Tide Generation at a Continental Shelf Break , LAURA BRANDT, JAMES ROTTMAN, KYLE BRUCKER, DOUGLAS DOMMERMUTH, Naval Hydrodynamics Division, Science Applications International Corporation — A fully nonlinear, three-dimensional numerical model is developed for the simulation of tidal flow over arbitrary bottom topography in an ocean with realistic stratification. The model is capable of simulating accurately at the generation of fine-scale internal wave tidal beams, their interaction with an ocean thermocline and the subsequent generation of solitary internal waves that propagate on this thermocline. Several preliminary simulation results are shown for uniform and non-uniform flow over an idealized two-dimensional ridge, which are compared with linear theory, and for flow over an idealized twodimensional continental shelf.

Tuesday, November 20, 2012 1:00PM - 3:10PM –

Session R2 Convection and Buoyancy-Driven Flows VIII 23A - Chair: Ralf Wittenberg, Simon Fraser University

1:00PM R2.00001 Spatial localization due to the interaction between convection and a large scale mode[1] , HSIEN-CHING KAO, EDGAR KNOBLOCH, UC Berkeley, Department of Physics — Spatially modulated states are of considerable interest in both rotating convection [1] and in magnetoconvection [2]. The formation of such states is due to the interaction between convective rolls and a large scale phase-like mode [3]: zonal velocity in rotating convection and magnetic potential in magnetoconvection. We have developed a higher order theory to describe the effects of spatial modulation near a certain codimension-two point where the leading order theory breaks down [1]. The theory leads to a fifth order Ginzburg-Landau equation with nonlocal terms. The properties of this equation are analyzed and the solutions used to explain the properties of spatially localized convectons in the full system determined numerically in [1,2].

[1] C. Beaume et al., submitted to J. Fluid Mech. (2012)
[2] D. Lo Jacono, A. Bergeon and E. Knobloch, J. Fluid Mech. 687, 595 (2011)
[3] S. M. Cox and P. C. Matthews, Physica D 149, 210229 (2001)

[1]This work was supported by National Science Foundation under grant DMS-0908102.

1:13PM R2.00002 Conservative bounds on heat transport in turbulent convection , RALF WITTENBERG, Simon Fraser University, JARED WHITEHEAD, CNLS, Los Alamos National Laboratory — The scaling dependence of the Nusselt number measuring heat transport in turbulent convection with the driving force remains incompletely understood, despite considerable effort in experiment, direct numerical simulation and theory. Variational upper bounds derived systematically from the governing partial differential equations provide a constraint on the possible scaling behaviors. We survey conservative analytical bounds on turbulent heat transport derived via the background flow method, both those obtained rigorously and semi-optimal upper bounds computed by numerical solution of the variational problem over a restricted class of backgrounds. We consider a range of scenarios, including the effects of plate conductivity, velocity boundary conditions and/or infinite Prandtl number in Rayleigh-Bénard convection, as well as related problems such as internal-heating-driven and porous medium convection.

1:26PM R2.00003 Localized structures in two-dimensional rotating convection , CEDRIC BEAUME, ALAIN BERGEON, IMFT, Universite de Toulouse (France), HSIEN-CHING KAO, EDGAR KNOBLOCH, Department of Physics, UC Berkeley, TOULOUSE TEAM, BERKELEY TEAM — Geophysical flows exhibit localized structures such as cyclonic and anticyclonic vortices. We consider here convection in a two-dimensional fluid layer with stress-free fixed temperature boundaries rotating uniformly about the vertical [1], and focus on steady spatially localized structures called convectons. These solutions are of two types, odd and even parity, and are found in both subcritical and supercritical regimes [2]. We describe the properties of these convectons and use numerical continuation in a periodic domain to show that the convecton branches exhibit behavior known as slanted snaking. The results are compared to weakly nonlinear theory [2,3].

[1] G. Veronis, J. Fluid Mech. 5, 401435 (1959)
[2] C. Beaume et al., preprint submitted to J. Fluid Mech. (2012)
[3] S. M. Cox and P. C. Matthews, Physica D 149, 210229 (2001)

1:39PM R2.00004 Pattern formation in nonlinear solutal Marangoni convection: three-dimensional simulations vs. experiments[1] , THOMAS KOELLNER, TU Ilmenau, Germany, KARIN SCHWARZENBERGER, KERSTIN ECKERT, TU Dresden, Germany, THOMAS BOECK, TU Ilmenau, Germany, MARANGONI INSTABILITY COLLABORATION — We present simulations and related experiments of the stationary solutal Marangoni convection. We performed three dimensional DNS of a 2-layer fluid-fluid system with surfactant transfer from one layer to the other. Our simulations successfully reproduced the diverse set of flow patterns, which was so far only observed in the experiments. The highly resolved simulations are performed with a specialized spectral method. The experimental system is modeled by two immiscible Newtonian fluids. Both fluids are separated by a plane interface. Initially, a surface active agent(surfactant) is dissolved in the upper phase. The purely diffusive transport of the surfactant is unstable to the stationary Marangoni instability due to variations of the interfacial solute concentration. The surfactant transport at the interface is modeled by Henry's law. The Schmidt number for the considered organic surfactant is usually much more than a thousand. The dynamics of the evolving patterns are described in detail and compared to experimental observation for a cyclohexanol/water system with butanol as transported solute. We classify the emerging structures and analyze their characteristic length scales in terms of the velocity and surfactant distribution.

[1]The work is funded under the frame of DFG priority program 1506.

1:52PM R2.00005 Weakly nonlinear stability of Marangoni convection in a liquid bridge[1] , KAORU FUJIMURA, Tottori University — Marangoni convection arising in a liquid column, bridging between concentric, circular parallel plates with different but uniform temperatures, is examined on its linear and weakly nonlinear stability. The analyses are conducted for small and moderate Prandtl numbers $0.001 \leq P \leq 10$. Our attention is focused on a relatively low liquid bridge with $h/r_0 = 1$ where h is the height and r_0 is the radius. The buoyancy effect is ignored and perfectly insulating condition is imposed on the surface of the liquid bridge. Linear stability analysis revealed that the critical condition was given by different azimuthal wavenumbers, $m = 1$, 2, and 3, depending on the Prandtl number. The critical condition is given by steady solutions for $0 < P < 0.0578$ and by oscillatory solutions for $P > 0.0578$. Weakly nonlinear analysis identifies the stable region of the secondary solutions bifurcating from the linear critical curve.

[1]Supported by Grant-in-Aid for scientific research (B) 21360101 and 2436007801.

2:05PM R2.00006 Transient diffusive boundary layers in porous media: optimal perturbations , DON DANIEL, NILS TILTON, AMIR RIAZ, University of Maryland, College Park — We study the linear stability of gravitationally unstable, transient, diffusive boundary layers in porous media using nonmodal stability theory. We perform a classical optimization procedure to obtain perturbations with maximum subsequent amplification. Due to the transient base-state, optimal perturbations depend on the initial perturbation time. At small times, optimal perturbations extend beyond the boundary layer producing unphysical initial conditions. To reciprocate experimental conditions, we propose a modified optimization procedure using an adjoint-based optimization formulation that constrains the initial perturbation within the boundary layer. Interestingly, dominant wavenumbers obtained using resultant perturbations exhibit different temporal behavior in comparison to the classical scheme. We validate our results using nonlinear direct numerical simulations.

2:18PM R2.00007 Transient diffusive boundary layers in porous media: The linear transition region , NILS TILTON, DON DANIEL, AMIR RIAZ, University of Maryland, College Park — Gravitationally unstable, transient, diffusive boundary layers play an important role in carbon dioxide sequestration in subsurface porous aquifers. Though the linear stability of these boundary layers has been studied extensively, there is little consensus concerning the critical time for instability. Nor is it clear which perturbations dominate the linear regime and trigger onset of convection due to nonlinear effects. We perform a comprehensive linear stability analysis using complementary quasi-steady and initial value problem approaches. We demonstrate that disagreement concerning the linear regime stems from an inherent sensitivity of the problem to how perturbation growth is measured. The perturbation concentration and velocity fields exhibit differing growth rates and these rates depend on the norm used to measure perturbation growth. Consequently, the critical time is not clearly defined. At later times, however, all initial perturbations tend towards the least stable quasi-steady eigenmode. We interpret this convergence process in terms of mechanisms related to the transient base-state, non-self-adjoint linear stability operator, and initial condition. Finally, we suggest potential paths for onset of convection which we demonstrate with direct numerical simulation.

2:31PM R2.00008 Evaporation dynamics of ethanol drops under terrestrial and reduced gravity levels[1] , FLORIAN CARLE, BENJAMIN SOBAC, DAVID BRUTIN, Aix-Marseille University - UMR 7343 IUSTI Laboratory — This experimental study, performed under microgravity conditions, focuses on the evaporation dynamics of ethanol drops and the formation and behaviour of the hydrothermal waves (HTWs) that spontaneously develop on the drops' surfaces. The aim of this study is to compare our results to a similar study performed under normal gravity conditions to confirm the purely thermocapillary origin of these instabilities. Under normal gravity conditions, a temperature gradient develops during the evaporation from the apex of the drop and the contact line, resulting in a gradient of surface tension, generating instabilities. HTWs flow radially around the apex where most of the evaporation takes place. In microgravity, the temperature gradient isn't as much defined as the one in normal gravity, but the apex maintains a temperature below the one of triple line. For different substrate temperatures and different levels of gravity, the HTWs follow a power law decay of the number of instabilities. Microgravity experiments show the same power law evolution. A scaling law succeed to predict with a good agreement the number of instabilities that form, regardless of the drop diameters, the substrate temperatures and the gravity levels.

[1]We would like to thank the "Centre National d'Études Spatiales" for their financial support for parabolic flights campaigns.

2:44PM R2.00009 Leidenfrost levitated liquid tori , STÉPHANE PERRARD, Matière et Systèmes Complexes, Université Paris Diderot, CNRS - UMR 7057, MATTHIEU LABOUSSE, EMMANUEL FORT, Institut Langevin, ESPCI ParisTech and Université Paris Diderot, CNRS UMR 7587, JOHN BUSH, MIT, YVES COUDER, LAURENT LIMAT, Matière et Systèmes Complexes, Université Paris Diderot, CNRS - UMR 7057 — A drop of water deposited on a surface hotter than 150°C can levitate without any contact with a solid container. Indeed the evaporation of the fluid generates a thin vapour film, which supports the drop's weight by lubrication forces (Leidenfrost effect). This effect was until now limited to droplets. We propose here an original substrate geometry, a circular brass through, that allows us to maintain in levitation any quantity of fluid. It could be a good tool to study wave propagation without solid boundary condition and thus very low friction. We report here one possible application, and our most striking observation : when the substrate temperature is high enough, convective motion appears in the liquid torus and its inner side becomes polygonal. This periodic deformation of large amplitude propagates along the azimuthal direction. The geometry, the flow and the shape appear very similar to the polygonal destabilization of an hydraulic jump. We propose here an experimental and theorical characterization of these rotating polygons having from three to twelve sides. Moreover, we have found a model describing the shape for any number of sides. It appears closely related to the Korteweg de Vries equation describing the propagation of solitonic waves in shallow water

2:57PM R2.00010 Hydrodynamic Instabilities Produced by Evaporation[1] , JULIO CESAR RUBEN ROMO-CRUZ, SERGIO HERNANDEZ-ZAPATA, GERARDO RUIZ-CHAVARRIA, Facultad de Ciencias, Universidad Nacional Autonoma de Mexico — When a liquid layer (alcohol in the present work) is in an environment where its relative humidity is less than 100 percent evaporation appears. When RH is above a certain threshold the liquid is at rest. If RH decreases below this threshold the flow becomes unstable, and hydrodynamic cells develop. The aim of this work is to understand the formation of those cells and its main features. Firstly, we investigate how the cell size depends on the layer width. We also study how temperature depends on the vertical coordinate when the cells are present. An inverse temperature gradient is found, that is, the bottom of liquid layer is colder than the free surface. This shows that the intuitive idea that the cells are due to a direct temperature gradient, following a Marangoni-like process, does not work. We propose the hypothesis that the evaporation produce a pressure gradient that is responsible of the cell development. On the other hand, using a Schlieren technique we study the topography of the free surface when cells are present. Finally the alcohol vapor layer adjacent to the liquid surface is explored using scattering experiments, giving some insight on the plausibility of the hypothesis described previously.

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

Tuesday, November 20, 2012 1:00PM - 3:10PM –

Session R3 Multiphase General II 23B - Chair: Sascha Hilgenfeldt, University of Illinois at Urbana-Champaign

1:00PM R3.00001 Light Field Imaging of Turbulent Liquid Sheet Breakup in Air , BARRY SCHARFMAN, ALEXANDRA TECHET, MIT, Dept. of Mechanical Engineering — The atomization of an unsteady turbulent sheet of water in air was analyzed using a combination of light field imaging (LFI) and synthetic aperture (SA) refocusing techniques. This sheet collides with and initially flows along a solid inclined plate, and imaging was performed in the region where breakup and separation from the plate begins. Ligaments and droplets emanate from the sheet and break off due to capillary instabilities. Image volumes consisting of these flow features, as well as segments of the liquid sheet body, were captured using a multiple CCD sensor array consisting of ten cameras arranged in three rows. Synthetic aperture refocusing techniques were 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. Feature shapes and spatial distributions have then been extracted from the refocused image volumes.

1:13PM R3.00002 Solid-Fluid flows using a variant of Immersed Boundary method in Gerris[1] , PEI SHUI, University of Edinburgh, UK, STÉPHANE POPINET, National Institute of Water and Atmospheric research, New Zealand, PRASHANT VALLURI, University of Edinburgh, UK, STÉPHANE ZALESKI, Institut Jean Le Rond d'Alembert, UMR7190 CNRS and Université Pierre et Marie Curie (Paris 6), France, MARTIN CRAPPER, University of Edinburgh, UK — An efficient 3D Immersed Boundary solver to simulate fully coupled fluid-solid interaction with 6 degrees-of-freedom (6DOF) solid movement enabling all translational and rotational motions has been developed in the GERRIS code. Here the solids are fully immersed in a flowing fluid driven either by pressure or shear. Solid objects of any arbitrary geometry and number can be considered. Prevention of overlap between the solids and wall is enforced through a repulsive force (Glowinski et al., 2001) which is the sum of all short-range interactions. The method agrees well with Stokes' settling and is validated against experiments published in literature. The simulation also agrees well with the classical case of a neutrally buoyant solid in shear flow and the orbit tracked by it (Jeffrey, 1922). 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 and will be presented in the conference. Comparison with experiments of Fortes et al (1987) concerning drafting, kissing and tumbling of two spherical solids during fluidization will also be presented.

[1]Richard Brown Fellowship, UPMC Paris

1:26PM R3.00003 On the mixture model of two-phase proppant transport in 1D fracturing flows , WEIMING LI, Halliburton — A mixture model of two-phase fluid flow is derived for proppant transport in 1D hydraulic fracturing. The governing equations of the model consists of the mass balance equations for the mixture and the proppant phase, the momentum equation for the mixture and the constitutive equation between proppant average velocity and fluid average velocity. Mass loss and momentum loss due to proppant settling are considered. One dimensional numerical simulations based on discontinuous Galerkin finite element method in space are performed. Both steady cases and transient cases are compared with available analytical solutions or manufactured solutions. Predicted numerical results agree well with exact solutions. This one dimensional two-phase model captures necessary proppant flow phenomena in hydraulic fracturing and also provides numerical efficiency and accuracy.

1:39PM R3.00004 A GPU-accelerated interfacial flow solver with advected normals: Application to contact line problems , ASHISH PATHAK, MEHDI RAESSI, University of Massachusetts Dartmouth — Accurate curvatures are essential for modeling surface tension forces in interfacial flows. The advected normals method by Raessi *et al.* yields curvatures that converge with mesh refinement. The same level of accuracy cannot be achieved with traditional approaches to calculating curvature, in which the gradient of the volume-of-fluid (VOF) or level-set functions are used. The flow solver used here is based on the VOF method and uses the two-step projection with GPU acceleration. In addition to curvature calculation, the advected normals are used for reconstructing the interface. The method has been successfully applied before to two-dimensional flow problems. We present here its extension to three dimensions. The PDE based extension of the vectors around the interface which was slow and computationally intensive in the original form has been accelerated by developing a GPU version of the extension algorithm. The advected normals method was also extended to handle contact line problems. The method has an advantage over traditional approaches to imposing contact angle. Since the normal vectors "communicate" with each other in this method, the effect of imposed contact angle is "felt" by the normal vectors in a wider region and is independent of grid resolution.

1:52PM R3.00005 Jump Conditions for the Stokes Equations with Discontinuous Viscosity and an Incompressible Interface with Singular Forces in 3D , DAVID SALAC, PRERNA GERA, University at Buffalo SUNY — Here the jump conditions for pressure and velocity are presented for two-phase Stokes and constant density Navier- Stokes flow with discontinuous viscosity across an incompressible interface with singular forces in three dimensions. This is necessary to accurately model systems such as vesicles or red blood cells. While jump conditions for incompressible interfaces and continuous viscosity have been published, this is the first demonstration of the jump conditions for the discontinuous viscosity situation. The derivation is based on the immersed interface method and appropriate local interface conditions. In addition to presentation of the jump conditions a simple analytic case has been created to verify the method and will be shown.

2:05PM R3.00006 A study of pressure-driven displacement flow of two immiscible liquids using a multiphase lattice Boltzmann approach[1] , PRASANNA REDAPANGU, Department of Chemical Engineering, Indian Institute of Technology Hyderabad, Yeddumailaram 502 205, Andhra Pradesh, India, PRATAP VANKA, Department of Mechanical Science and Engineering, University of Illinois at Urbana-Champaign, USA, KIRTI SAHU, Department of Chemical Engineering, Indian Institute of Technology Hyderabad, Yeddumailaram 502 205, Andhra Pradesh, India — The pressure-driven displacement of two immiscible fluids in an inclined channel in the presence of viscosity and density gradients is investigated using a multiphase lattice Boltzmann approach. The effects of viscosity ratio, Atwood number, Froude number, capillary number and channel inclination are investigated through flow structures, front velocities and fluid displacement rates. Our results indicate that increasing viscosity ratio between the fluids decreases the displacement rate. We observe that increasing the viscosity ratio has a non-monotonic effect on the velocity of the leading front; however, the velocity of the trailing edge decreases with increasing the viscosity ratio. The displacement rate of the thin-layers formed at the later times of the displacement process increases with increasing the angle of inclination because of the increase in the intensity of the interfacial instabilities. Our results also predict the front velocity of the lock-exchange flow of two immiscible fluids in the exchange flow dominated regime.

[1]Department of Science and Technology, India

2:18PM R3.00007 Double diffusive effects between two miscible fluid flows in a channel , MANORANJAN MISHRA, Indian Institute of Technology Ropar, India, ANNE DE WIT, Université Libre de Bruxelles, Brussels, Belgium, KIRTI CHANDRA SAHU, Indian Institute of technology Hyderabad, India — The pressure-driven displacement flow of a less viscous fluid by a more viscous one in a horizontal channel is a stable configuration in the context of single component flows. However, we have shown by numerical simulations based on finite volume approach that the double-diffusive (DD) effects can destabilize this stably stratified system. Such effects can appear if the fluid consists of a solvent containing two solutes both influencing the viscosity of the solution and diffusing at different rates. The continuity and Navier-Stokes equations coupled to two convective-diffusion equations for the evolution of the concentration of the solutes are solved. The viscosity is assumed to depend on the concentration of both solutes, while density contrast is neglected. The results demonstrate the development of various new instability patterns in the presence of DD effects at the miscible "interface" separating the fluids. It is found that the intensity of the instability increases with increasing the diffusivity ratio of the solutes. This in turn increases the fluid mixing and accelerates the displacement of the fluid originally filled inside the channel.

2:31PM R3.00008 Microscopic aspects of Liquid Foam Fracture , SASCHA HILGENFELDT, Mechanical Science and Engineering, University of Illinois at Urbana-Champaign, PETER STEWART, OCCAM Mathematical Institute, The University of Oxford, STEPHEN DAVIS, Engineering Sciences and Applied Mathematics, Northwestern University — A layer of foam bubbles between parallel plates (a quasi-two-dimensional liquid foam) gives unique access to the details of microscopic configurations in a system whose macroscopic properties include both liquid and solid behavior. The failure of cohesion under stress in such a layer of bubbles offers a study case on fracture, which we have experimentally shown to occur both in a mode similar to fluid fingering and a mode similar to the cleavage of a solid material. Simulations elucidate the microscopic aspects and the fluid-dynamical mechanisms behind these processes, spanning a wide variety of length and time scales and incorporating both fracture modes. Aspects of microstructure and rate dependence can thus become part of a detailed study of fundamental fracture processes.

2:44PM R3.00009 Multiphase flow of miscible liquids: drops and jets , TRAVIS WALKER, ALISON LOGIA, GERALD FULLER, Stanford University — Drops and jets of liquids that are miscible with the surrounding bulk liquid are present in many processes. Although the interactions of immiscible drops and jets show similarities to miscible systems, the small, transient interfacial tension associated with miscible systems create distinct outcomes such as intricate droplet shapes, break-up resistant jets, and spreading sessile drops. Experiments have been conducted to understand several basic multiphase flow problems involving miscible liquids including the free-surface pendant drops resulting from drop impaction and the dissolution of sessile drops in a miscible bath. Using high-speed imaging of the morphological evolution of the flows, we show that these processes are controlled by interfacial tensions. Further multiphase flows include investigating miscible jets, which allow the creation of fibers and tubular shapes from inelastic materials that are otherwise difficult to process due to capillary breakup. This work shows that stabilization from the diminishing interfacial tensions of the miscible jets allow various elongated morphologies to be formed. When combined with a mechanism to freeze these fibers, highly oriented materials can be created.

2:57PM R3.00010 Numerical Study of Crossflow Enhanced Microfiltration of Oil-in-Water Emulsions , TOHID DARVISHZADEH, Graduate Student, Department of Mechanical Engineering, Michigan State University, NIKOLAI PRIEZJEV, Assistant Professor, Department of Mechanical Engineering, Michigan State University, VOLODYMYR TARABARA, Associate Professor, Department of Civil and Environmental Engineering, Michigan State University — The effective separation of dilute oil-in-water mixtures involves high flux of water through a porous membrane while maintaining high rejection rate of the oil phase. In this study, the effects of transmembrane pressure and crossflow velocity on rejection of oil droplets and thin oil films by pores of different cross-section are investigated numerically by solving the Navier-Stokes equation. We found that the presence of crossflow increases the efficiency of microfiltration by sweeping the dispersed phase away from the pore entrance at the membrane surface and thus enhancing overall water flux. With further increasing crossflow velocity, however, the shape of the droplet becomes strongly deformed near the pore entrance; and, at sufficiently high transmembrane pressures, the droplet breaks up into two fragments, one of which penetrates into the pore. The dynamics of an oil droplet near the pore entrance and the critical pressure of permeation are studied as a function of the oil viscosity, ratio of drop to pore radii, surface tension, and contact angle.

Tuesday, November 20, 2012 1:00PM - 2:31PM –

Session R4 Foams 23C - Chair: Peter Stewart, University of Oxford

1:00PM R4.00001 Surface waves in a foam , ANNE LE GOFF, MMN, UMR Gulliver 7083, ESPCI, PABLO COBELLI, DF, FCEN, UBA & IFIBA, CONICET, Argentine / PMMH, UMR 7636, ESPCI, France, GUILLAUME LAGUBEAU, USACH, Santiago, Chile / LAUM, Université du Maine, France — We investigate the propagation and attenuation of waves generated at the surface of a liquid foam after the impact of a solid sphere. Surface deformation is recorded with high spatial and temporal resolution thanks to a fringe projection technique. We show that most surface waves travel at a velocity of a few meters per second. High velocity impacts also trigger the emission of faster waves. We discuss the nature of these two types of waves.

1:13PM R4.00002 The effect of gravity on drainage and rupture in surfactant-free foams[1] , MICHAEL DAVIS, Northwestern University, PETER STEWART, Oxford University, STEPHEN DAVIS, Northwestern University — In low liquid-fraction surfactant-free foams, lamella thinning due to drainage leads to rupture by van der Waals instability, which causes coarsening due to coalesence of neighboring bubbles. We use asymptotic analysis to predict the effect of gravity on drainage flow for a flat, vertical lamella and for a weakly bent, horizontal lamella. In both cases the films thin non-uniformly, and the thinning is exponential for long times; much faster than the power law thinning rate predicted for gravity-free lamella drainage. The asymptotic solutions are also able to predict the onset of rupture in both geometries. Numerical solutions are used to verify these asymptotic predictions, and to determine their range of validity for relevant parameters.

[1]Work supported by NSF-CMMI 0826703.

1:26PM R4.00003 Coalescence driven coarsening in surfactant-free foams[1] , PETER S. STEWART, The University of Oxford, STEPHEN H. DAVIS, Northwestern University — We consider the stability of a planar gas-liquid foam with low liquid fraction, in the absence of surfactants and stabilising particles, as a model for molten metal foams produced as a precursor to forming high-porosity metallic solids. We adopt a network modelling approach, treating the bubbles as polygons, the accumulation of liquid at the bubble vertices as dynamic nodes and the liquid bridges separating the bubbles as uniformly thinning free films. We further incorporate an explicit rupture criterion for the films once they become sufficiently thin, due to van-der-Waals intermolecular attractions. We initialise the foam as a mono-disperse array of regular hexagonal bubbles, and examine the rate of coarsening as the films break and the bubbles rapidly coalesce.

[1]National Science Foundation Grant No. CMMI-0826703

1:39PM R4.00004 Particle-tracking velocimetry analysis of liquid drainage within individual Plateau borders in aqueous foam , MATTHEW J. KENNEDY[1], MICHAEL W. CONROY, RAMAGOPAL ANANTH, JAMES W. FLEMING, Naval Research Laboratory — Foam drainage theory describes macro-scale liquid drainage for a body of foam based on the microscopic flow within individual Plateau borders and within the nodes which occur at the intersections of multiple Plateau borders. The present study measures micro-scale liquid velocities within individual Plateau borders using microparticle image velocimetry, and it measures macro-scale liquid drainage using a weighing scale. Measurements take place over the course of free drainage for foam which is initially wet with initial liquid fraction equal to 20% averaged over the height of the foam. Preliminary results show that the flow dynamics within individual Plateau borders evolve according to similar trends as the macro-scale volume of liquid drained. Foam drainage theory agrees with both measurements after an initial transition period, but during initial drainage the experimentally measured drainage rate exceeds that predicted by the theory. We discuss implications of the agreement between the micro-scale and macro-scale measurements as well as potential sources for the unexpectedly high drainage rate which occurs at the beginning of drainage.

[1]NRC Research Associate at NRL

1:52PM R4.00005 Self healing: solid spheres impacting soap bubbles , TAYLOR KILLIAN, JOSHUA BRYSON, JORDAN HUEY, Brigham Young University, JAMES C. BIRD, Boston University, JEAN-CHRISTOPHE NAVE, McGill University, TADD TRUSCOTT, Brigham Young University — Under the right conditions a moving sphere may pass through a stationary soap bubble without rupturing it. At impact, the sphere forms a cavity in the soap film that often facilitates reparation after collapse. This interaction leaves a small film surrounding the sphere as it passes through the center of the bubble. In contrast, as the sphere passes through the opposite side of the bubble, rupture is more likely. The physics behind this phenomenon are not well understood, nor the limiting factors of this interaction. We explore the phenomenon using high-speed photography. Our observations reveal that there are several distinct cavity regimes. We present the parameters for drainage, rupture and reparation each of which are related to curvature gradients.

2:05PM R4.00006 Numerical Modeling of Nanocellular Foams Using Classical Nucleation Theory and Influence Volume Approach , IRFAN KHAN, STEPHANE COSTEUX, SHANA BUNKER, JONATHAN MOORE, KISHORE KAR, The Dow Chemical Company — Nanocellular porous materials present unusual optical, dielectric, thermal and mechanical properties and are thus envisioned to find use in a variety of applications. Thermoplastic polymeric foams show considerable promise in achieving these properties. However, there are still considerable challenges in achieving nanocellular foams with densities as low as conventional foams. Lack of in-depth understanding of the effect of process parameters and physical properties on the foaming process is a major obstacle. A numerical model has been developed to simulate the simultaneous nucleation and bubble growth during depressurization of thermoplastic polymers saturated with supercritical blowing agents. 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 model is modified to include mechanisms for Joule-Thompson cooling during depressurization and secondary foaming. Simulation results for polymer with and without nucleating agents will be discussed and compared with experimental data.

2:18PM R4.00007 A novel method of producing stable emulsions via electrified W/O interfaces , BEHNAM SADRI, University of Alberta, PEJMAN TABATABAEE-HOSSEINI, BABAK VAJDI HOKMABAD, MEHDI REZAYATI CHARAN, ESMAEIL ESMAEILZADEH, University of Tabriz, MULTIPHASE FLOW GROUP TEAM — In the current paper a vertical electric field was induced to the liquid interface to make thin jets from the conical tip structures. By the means of this jet dispersion a novel method of emulsification of water drops in dielectric medium was represented. Experiments reported in this paper enable a comprehensive illustration of introduced mechanism for the emulsification. The important aspects of an emulsion production were investigated through produced droplets properties, including their movement velocity, and size distribution. Variation of mentioned parameters was investigated with the conductivity throughout various electrolyte ion concentration additions to the dispersed phase. Experiments show that, conductivity augmentation antedates the stable cone-jet formation to the lower electric fields. Furthermore, it reduces the stability duration of emulsified drops due to increasing in polydispersity and coalescence rate.

Tuesday, November 20, 2012 1:00PM - 3:10PM –

Session R5 Computational Fluid Dynamics VIII 24A - Chair: Hamid Rahai, California State University, Long Beach

1:00PM R5.00001 A high order solver for the unbounded Poisson equation with specific application to the equations of fluid kinematics , M.M. HEJLESEN, J.T. RASMUSSEN, Department of Mechanical Engineering, Technical University of Denmark, Building 403, DK-2800 Kgs. Lyngby, Denmark, P. CHATELAIN, Institute of Mechanics, Materials and Civil Engineering, Universite catholique de Louvain, B-1348, Belgium, J.H. WALTHER[1], Department of Mechanical Engineering, Technical University of Denmark, Building 403, DK-2800 Kgs. Lyngby, Denmark — This work improves upon Hockney and Eastwood's Fourier-based algorithm for the unbounded Poisson equation to formally achieve an arbitrary high order of convergence. The high order convergence is achieved by constructing regularized Green's functions through a filtering procedure. High order filters and regularized kernels are obtained by canceling the corresponding moments, a task which we show can be performed through a recursive application of extrapolation. We assess the methodology on the kinematic relations between the velocity and vorticity fields. In fluid mechanics the velocity is determined from the curl of the stream function which in turn is related to vorticity through the Poisson equation. The curl operator can be computed analytically, as a multi-component convolution kernel or, alternatively, computed directly in Fourier space through a spectral differentiation of the convolution product. The latter solution allows to reduce the computational cost and memory footprint of the algorithm while conserving its convergence order.

[1]Also at: Computational Science and Engineering Laboratory, ETH Zurich, Universitatsstrasse 6, CH-8092, Zurich, Switzerland

1:13PM R5.00002 A fully spectral efficient algorithm for Stokes suspension simulations in doubly periodic confined geometries , JAE SUNG PARK, Department of Chemical and Biological Engineering, University of Wisconsin-Madison, DAVID SAINTILLAN, Department of Mechanical Science and Engineering, University of Illinois at Urbana-Champaign — The calculation of hydrodynamic interactions between suspended particles under confinement in low-Reynolds number flows is a computationally expensive task. In this study, we develop an efficient spectral method for the calculation of these interactions in a doubly periodic geometry between two infinite parallel no-slip walls in Stokes flow, a geometry commonly encountered in microfluidic devices. We consider the flow generated by a distribution of point forces inside a unit cell, and decompose it into the sum of two Stokes problems. The first one involves triply periodic boundary conditions and makes use of our previously developed fast smooth particle-mesh Ewald algorithm, while the second one provides a correction to the periodic solution to satisfy the no-slip boundary condition on the confining walls. This second problem is based on an analytic solution for the flow between two flat plates with prescribed Dirichlet wall boundary conditions that are determined from the first problem, and the total solution can be then obtained by superimposing the solution of each problem using the linearity of the Stokes equations. We conclude by presenting an application of this method to a confined suspension of spherical particles.

1:26PM R5.00003 Multithreaded Impliclty Dealiased Convolutions for Pseudospectral Simulations , MALCOLM ROBERTS, Aix-Marseille University, JOHN C. BOWMAN, University of Alberta — Convolutions form the crux of the pseudospectral method for direct numerical simulations of nonlinear PDEs such as the Navier–Stokes equations and magnetohydrodynamic flows. The computation of convolutions is an expensive task that is facilitated by the use of the convolution theorem and FFTs. However, input data must be zero-padded in order to remove aliased terms and recover a linear convolution. Here, we present a multithreaded version of the method of implicit dealiasing (Bowman and Roberts, SIAM J. Sci. Comput. 33, 2011). Implicit dealiasing has computational complexity identical to the conventional zero-padding technique, but is twice as fast in practice and requires $(2/3)^{d-1}$ the memory of a conventional d-dimensional centred convolution. High-performance implicitly dealiased convolution routines are available under the LGPL at `fftwpp.sourceforge.net`.

1:39PM R5.00004 A spectral multi domain decomposition method for computing the 2D backward facing step flow , ARJUN JAGANNATHAN, MANHAR DHANAK, RANJITH MOHAN, Florida Atlantic University — A Chebyshev spectral domain decomposition method is developed for computing the characteristics of the 2D incompressible backward facing step flow at low to moderate Reynolds numbers. A step to channel height of 1:2 and an inlet channel of length twenty times the step height are chosen for the study. A total of five sub domains is used and an influence matrix technique is employed for carrying out the domain decomposition. The unsteady 2D Navier Stokes equations are solved in the vorticity-streamfunction formulation with an explicit second order Adams Bashforth time marching scheme. At the inlet, a parabolic velocity profile is initialized and the outflow boundary is located sufficiently far away from the step so that for the particular Reynolds numbers studied, the parabolic velocity profile is retrieved at the outlet. A non-reflecting boundary condition (cf. Jin and Braza, 1993) wherein we set the elliptic $(\partial^2/\partial x^2)$ terms in the governing equations to zero at the outflow boundary is found to work well for this purpose. Detailed steady state results for Reynolds numbers in the range 100 to 800 are presented and compared with other numerical and experimental results found in the literature.

1:52PM R5.00005 A deformed spectral quadrilateral multi-domain penalty model for the incompressible Navier-Stokes equations[1] , SUMEDH JOSHI, Center for Applied Mathematics, Cornell University, PETER DIAMESSIS, Department of Civil and Environmental Engineering, Cornell University — A penalty method is a variant of a spectral element method that weakly enforces continuity between adjacent elements and weakly enforces continuity at physical boundaries. Furthermore, at the boundaries, the PDE is also partially satisfied. The spirit of such a formulation is that in theory, a PDE operates arbitrarily close to any measure-zero boundary. Here, a previous spectral multi-domain penalty model for the incompressible Navier-Stokes equations is extended to include deformed boundaries for shoaling-type problems encountered in environmental fluid mechanics. Some difficulties addressed include satisfying compatibility conditions in a (pseudo-)pressure Poisson equation that arises. A previous strategy to satisfy compatibility by use of a null singular vector is presented and strategies to enforce compatibility for the deformed problem are discussed. Results are shown for standard incompressible flow benchmarks. The primary goal of this work is to model nonlinear internal wave propagation along a shallow, sloping bathymetry, as may be characteristic of a continental shelf region.

[1]Work supported by the NSF and the NDSEG fellowship.

2:05PM R5.00006 Direct Numerical Simulation of Compressible Turbulent Flows with Weighted Non-Linear Compact Schemes , DEBOJYOTI GHOSH, SHIVAJI MEDIDA, JAMES BAEDER, University of Maryland — The numerical solution of compressible, turbulent flows requires a high-resolution, non-oscillatory algorithm to resolve a large range of length scales. Conventional non-linear flux limited schemes are too dissipative for length scales relevant to turbulent flow features, while compact interpolation schemes with high spectral resolution require monotonicity-preserving filtering for flows with discontinuities. The Compact Reconstruction WENO (CRWENO) scheme (*Ghosh and Baeder, SIAM J. Sci. Comput., 34(3), 2012*) uses a non-linear, solution-dependent combination of low-order compact interpolation schemes to yield a high-order accurate, non-oscillatory reconstruction scheme with high spectral resolution. Previous studies by the authors have demonstrated the improved performance of the CRWENO scheme at preserving and resolving smooth and discontinuous flow features (for one- and two-dimensional flow problems), compared to the WENO scheme of the same order of convergence. In the present study, the CRWENO scheme is applied to the direct numerical simulation of benchmark turbulent flow problems. In particular, the decay of isotropic turbulence and the shock-turbulence interactions are studied and the results are presented. The solutions from the CRWENO scheme are compared with those in the literature, obtained using WENO schemes and compact schemes with filtering.

2:18PM R5.00007 DNS of homogeneous turbulent shear flow using a hybrid Pseudospectral-WENO Method , PARVEZ SUKHESWALLA, T. VAITHIANATHAN, LANCE COLLINS, Cornell University — Pseudospectral-based direct numerical simulations (DNS) of homogeneous turbulent shear flow (HTSF) have been shown to inevitably suffer from numerical resolution problems that become more severe with increasing Reynolds number [Sukheswalla et al., in review]. The resulting Gibbs oscillations can be removed using low-pass spectral filters that stabilize the simulations and enable attainment of higher asymptotic Reynolds numbers. However, while low-pass filtering does not appear to impact large-scale statistics, it does compromise small-scale statistics such as vorticity, with unclear consequences on the overall dynamics over time. In this presentation, we put forth an alternative approach based on a hybrid DNS method, wherein a Weighted Essentially Non-oscillatory (WENO) scheme is used to compute the nonlinear convective term that is the primary source of the Gibbs oscillations, while a pseudospectral method is used for the other terms. The resulting hybrid scheme yields large- and small-scale statistics in good agreement with the experiments of Isaza et al. [*Phys. Fluids* 21(6), 2009] and Gylfason et al. [*J. Fluid Mech.* 501:213–229, 2004]. The effectiveness of the new scheme for the fluid velocity lays the groundwork for future DNS of inertial particles in HTSF.

2:31PM R5.00008 A High Order Volume Penalty Method , DAVID SHIROKOFF, JEAN-CHRISTOPHE NAVE, McGill University — The volume penalty method provides a simple, efficient approach for solving the incompressible Navier-Stokes equations in domains with boundaries or in the presence of moving objects. Despite the simplicity, the method suffers from poor convergence in the penalty parameter, thereby restricting accuracy of any numerical method. We demonstrate that one may achieve high order accuracy by altering the form of the penalty term. We discuss how to construct the modified penalty term, and provide 2D numerical examples demonstrating improved convergence for the heat equation and Navier-Stokes equations.

2:44PM R5.00009 Assessing the Recovery-based Discontinuous Galerkin Method for Turbulence Simulations , ADITYA NAIR, ERIC JOHNSEN, SREENIVAS VARADAN, University of Michigan, Ann Arbor — The Discontinuous Galerkin (DG) method offers significant advantages over traditional finite difference and finite volume methods, such as high parallel scalability, portability to complex geometries and super-convergence. However, DG has yet to emerge as a viable option for turbulence simulations, due to the lack of a consistent and accurate diffusion scheme. Currently, orders of $p+1$ are achieved, where p is the polynomial order within a cell. A promising approach is that of recovery, which has been shown to exhibit convergence rates up to $3p+2$ in one dimension. This technique is based on the idea of enhanced recovery, where the underlying solution is recovered over neighboring cells and appropriately enhanced in the face-tangential directions. We use several test problems (pure diffusion, Taylor-Green vortex) to show that we achieve the same convergence rates in multiple dimensions, and compare this approach to other common diffusion schemes.

2:57PM R5.00010 Approach for robustly simulating supercritical fluid mixing with large density contrast using high-order schemes , HIROSHI TERASHIMA, MITSUO KOSHI, The University of Tokyo — We present a robust yet efficient approach for simulating supercritical fluid mixing with large density contrast using a high-order central differencing scheme. The present method is designed to maintain the pressure and velocity equilibriums at fluid interfaces for any type of discretization and equation of state: the pressure evolution equation is introduced for the pressure equilibrium and the numerical diffusion terms for the mass and momentum equations are consistently constructed for the velocity equilibrium. Thus, spurious oscillations possibly generated at interfaces are prevented, enabling robust applications of high-order schemes to severe thermodynamic fluid conditions. The consistent numerical diffusion term is also constructed for the species-mass conservation equation. Several examples of supercritical fluid problems such as interface advection and jet mixing problems demonstrate the robustness and superiority of the present method over a conventional conservative method.

Tuesday, November 20, 2012 1:00PM - 2:44PM –

Session R6 Microfluidics: Capillary III 24B - Chair: Shriram Pillapakkam, Temple University

1:00PM R6.00001 Trapping and release of bubbles from a linear pore , ANNE JUEL, GEOFFREY DAWSON, University of Manchester, UK, SUNGYON LEE, UCLA — Multiphase flows of practical interest are characterized by complex vessel geometries ranging from natural porous media to man-made lab-on-a-chip devices. Models based on the over-simplification of the pore geometry often suppress fundamental physical behavior. We study the effect on bubble motion of a sudden streamwise expansion of a square tube. The extent to which a bubble driven by constant flux flow broadens to partially fill the expansion depends on the balance between viscous and surface tension stresses, measured by the capillary number Ca. This broadening is accompanied by the slowing and momentary arrest of the bubble as Ca is reduced towards its critical value for trapping. For $Ca < Ca_c$ the pressure drag forces on the quasi-arrested bubble are insufficient to force the bubble out of the expansion so it remains trapped. We examine the conditions for trapping by varying bubble volume, flow rate of the carrier fluid, and length of expanded region, and find that Ca_c depends non-monotonically on the size of the bubble. We verify with experiments and a capillary static model that a bubble is released if the work of the pressure forces over the length of the expansion exceeds the surface energy required for the trapped bubble to reenter the constricted square tube.

1:13PM R6.00002 Direct measurements of air layer profiles under impacting droplets using high-speed color interferometry , ROELAND VAN DER VEEN, TUAN TRAN, DETLEF LOHSE, CHAO SUN, Physics of Fluids Group, Faculty of Science and Technology, University of Twente, 7500AE Enschede, The Netherlands — A drop impacting on a solid surface deforms before the liquid makes contact with the surface. We directly measure the time evolution of the air layer profile under the droplet using high-speed color interferometry, obtaining the air layer thickness before and during the wetting process. Based on the time evolution of the extracted profiles, we measure the velocity of air exiting from the gap between the liquid and the solid, and account for the wetting mechanism and bubble entrapment. We determine the entrained bubble volume for varying impact velocities using the air layer profiles and compare it to theory and numerical simulations. The present work offers a tool to accurately measure the air layer profile and quantitatively study the impact dynamics at a short time scale before impact.

1:26PM R6.00003 Encapsulating Ellipsoids in Drops[1] , MICHAEL NORTON, TERESA BRUGAROLAS, JONATHAN CHOU, HAIM BAU, DAEYEON LEE, University of Pennsylvania — Large aspect ratio particles were produced by embedding spherical polystyrene particles within a polymer film and subsequently heating and stretching the film. Particles were released by dissolving the film. Using a flow-focusing device, the elongated particles were partially encapsulated within droplets of fluid A, such as water, surrounded by an immiscible fluid B, such as oil. Drop volumes were controlled by adjusting the flow rates of fluids A and B. The contact angle was adjusted indirectly by varying the amount of surfactant adsorbed to the particle surface. The encapsulation process was visualized with a high-speed video camera. We observed cases ranging from partial to complete encapsulation and examined experimentally and theoretically the shape of the interface between fluid A and fluid B as a function of the drop volume. The numerically predicted position of the pinning line and the shape of the drop were compared to experimentally produced conformations and agreed favorably.

[1]This work was supported by ITMAT (UL1RR024134 from the NCRR) and the Penn MRSEC (NSF DMR-1120901)

1:39PM R6.00004 Experimental study of adsorption of particles at two-fluid interfaces , VADIM LINEVICH, SHRIRAM PILLAPAKKAM, Temple University, PUSHPENDRA SINGH, New Jersey Institute of Technology — The inertia of a particle plays an important role in its motion in the direction normal to a two-fluid interface, and in determining its adsorption trajectory. Although the importance of inertia diminishes with decreasing particle size, on an air-water interface the inertia continues to be important even when the particle size is as small as few nanometers. We have recently shown that the motion of particles in the direction normal to the interface while being adsorbed gives rise to a secondary lateral flow on the interface that causes newly adsorbed particles to disperse. The goal of this study is to experimentally study the adsorption trajectory of particles at two-fluid interfaces. Specifically, we will analyze the effect of particle size on the adsorption trajectory of a single particle, as it is being trapped at an air-water interface. The diameter of a particle will be varied between $\sim$ 100 - 1000 microns. Our experimental setup consists of a high speed camera with a recording speed up to 3000 frames per second which is connected to a horizontal microscope. Captured high speed videos are analyzed to determine the frequency and the amplitude of oscillation during adsorption.

1:52PM R6.00005 Dissipative particle dynamics simulation of a liquid meniscus confined between atomic force microscope tip and substrate , ZHEN LI, CHUANJIN LAN, YANBAO MA, School of Engineering, University of California, Merced — Liquid meniscus forms between the atomic force microscope (AFM) tip and the substrate under ambient humidity. The liquid meniscus affects the AFM measurements and plays an important role in dip-pen nanolithography. To understand the behaviors of the meniscus, a mesoscopic methodology called dissipative particle dynamics (DPD) is utilized to investigate the liquid meniscus confined between AFM tip and a solid surface. Results show that the structure of the liquid meniscus is highly dependent on the wettability properties of the tip and the substrate as well as the tip-to-surface distance. The area of liquid-solid interface increases as the wetting properties of the tip and substrate change from hydrophilic to hydrophobic, which results in a transition of the meniscus shape from convex to concave. The wetting properties of solid surface affect the process of the liquid meniscus breakup as the tip-to-surface distance increase. This nonlinear process is also affected by the surface tension of the liquid, thermal fluctuation and the speed of tip.

2:05PM R6.00006 Capillary-Driven Flow through Optimal Wick Structures for Heat Pipe Applications , YU-WEI LIU, MARIN SIGURDSON, CARL MEINHART, UCSB — In this study we investigate surface tension-driven fluid motion through the wick structure of a flat heat pipe. Wick designs in flat heat pipes are typically limited by viscous drag and capillary pressure, and do not transport fluids at sufficient rates to meet high-demand cooling requirements. An analytical model is used to describe flow through wick structure with array of channels. The capillary pressure and viscous drag are obtained by calculating surface energy difference and solving Stokes equations, respectively. To verify the model, we conducted wetting tests on the wick samples with different channel dimensions. The model agrees qualitatively with the experiments, but under predicts the viscous drag, which is due to the meniscus effect at the surface. The concave liquid surface increases the pressure drop, and therefore reduces the effective permeability. To include the meniscus effect, we combined Stokes equations with deformed surface in a full 3D simulation by COMSOL V4.2a (COMSOL, Inc., Stockholm, Se). The Young-Laplace law and the curvature equation are used to determine the deformed surface.

2:18PM R6.00007 Droplets interacting with structured microchannel walls , RIELLE DE RUITER, MICHEL DUITS, FRIEDER MUGELE, University of Twente — The presence of water droplets in oil-filled microfluidic systems influences the effective resistance of the microchannels. We study the effect of droplets interacting with topographically structured walls, for example single defects or a periodic array of grooves. Depending on the specific geometry and flow conditions, the droplet is trapped in or released from the structured region. Using a so-called microfluidic comparator (a microfluidic unit to compare flow rates in two channels), we quantify the effect of the presence of the droplet and its deformation by the flow on the effective resistance of the microchannel. These insights can be used to optimize the release of entrapped oil during enhanced oil recovery.

2:31PM R6.00008 Lattice Boltzmann Simulations of Finite-Sized Particles in Interfaces , KEVIN CONNINGTON, The Levich Institute, The City College of New York, TAEHUN LEE, Mechanical Engineering, The City College of New York, JEFF MORRIS, The Levich Institute, The City College of New York — The presence of solid particles can play an important role in many multiphase/multi-component flows. For example, particles in the interface of an emulsion can act to stabilize the drop, preventing breakup. We extend an existing free energy-based multi-component Lattice Boltzmann Method (LBM) to handle the transport of immersed solid particles of a finite size. The multi-component LBM can simulate the property differences encountered in a water-air system while eliminating the unwanted phenomenon of spurious currents at equilibrium. The particles are transported by the fluid according to Newtonian dynamics. The total force on a particle is computed by Momentum Exchange (ME), as in single phase flow. However, we introduce a supplemental term to account for the force of the interface on the particle. We validate the inclusion of this forcing term, and demonstrate the capability of the code by performing simulations of drop impact with immersed solid particles and the rupture of a liquid bridge containing particles.

Tuesday, November 20, 2012 1:00PM - 3:10PM –

Session R7 Geophysical: General VI 24C - Chair: John Taylor, University of Cambridge

1:00PM R7.00001 Mixing efficiency of turbulent stratified flows: Not all flows are created equal[1] , ALBERTO SCOTTI, BRIAN WHITE, UNC — Small scale mixing in the stratified interior of the ocean is a fundamental, yet poorly characterized, controlling factor of the global Meridional Overturning Circulation (MOC). In the oceanographic community, the mixing efficiency is usually assumed to be 20%. In this talk, we use DNS datasets to calculate the mixing efficiency in different class of flows. The mixing efficiency is calculated using the actual irreversible diapycnal flux of buoyancy (Winters and D'Asaro, 1996; Scotti et al., 2006) instead of the more customary turbulent diapycnal fluxes. This avoids potential issues of contamination of the latter from reversible processes (e.g., internal waves). For flows in which mixing is solely mechanically driven our profiles of mixing efficiency vs. turbulent intensity parameter agree well with the profiles previously established in the literature. However, for flows in which mixing is driven in part or fully by a thermodynamically induced excess of available potential energy, we obtain profiles characterized by much higher values of mixing efficiencies. Applications of these results to the MOC are discussed. Note: The DNS datasets of turbulent stratified channel flow was provided courtesy of M. Garcia-Villalba and J. C. del Alamo.

[1]Work sponsored by NSF and ONR

1:13PM R7.00002 Instability Mechanisms in a Stratified and Rotating Shear Layer with Horizontal Shear , ERIC AROBONE, SUTANU SARKAR, University of California, San Diego — One of the least understood scales of the ocean is the submesoscale. Here, rotation is important but does not necessarily control the dynamics. Instabilities and nonlinear cascades are possible in this regime while the influence of stable stratification is significant. Previous work by the authors revealed augmentation of the vertical wavenumber band associated with linear barotropic instability near the state of zero centerline absolute vorticity for strongly stratified flow. Enstrophy budgets from nonlinear simulations show a marked transition corresponding to the changing in sign of centerline absolute vorticity. We performed numerical experiments to examine the effect of both rotation and stratification on coherent dynamics with environmental parameters appropriate for submesoscale flows. Coherent structure evolution will be explored to understand the effect of stratification and rotation, in combination, on possible instability mechanisms, e.g. elliptic, zigzag, inertial, and barotropic instabilities. Additionally, physical mechanisms driving the flow evolution will be discussed with the aid of enstrophy budgets and flow visualizations.

1:26PM R7.00003 Transition in Energy Spectra and Vortex Structures in Stably Stratified Turbulence , YOSHIFUMI KIMURA, Graduate School of Mathematics, Nagoya University, JACKSON HERRING, NCAR — The power-law transition in the energy spectrum and relating vortex formation for stably stratified turbulence are investigated using the pseudo-spectral DNS of the Navier-Stokes equation under the Boussinesq approximation with 2048^3 grid points. From the zero total energy initial condition, integrated horizontal forcing is imposed in a narrow wave number band centered at k=5. At the first stage of the development, the horizontal energy spectra show a steep power-law ($\sim k_\perp^{4-5}$). By this time, we observe that many wedge vortices are produced and they move horizontally (like dipoles) in random directions. This stage lasts a long period of time, and then the tail part of the spectrum begins to rise to show the Kolmogorov-type slope ($k_\perp^{-5/3}$). During this stage of time, 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 local Richardson number small to develop Kelvin-Helmholtz billows. The relation between the horizontal breaking of the Kelvin-Helmholtz billows and the observation of the Kolmogorov-type slope will be discussed.

1:39PM R7.00004 Large Eddy Simulations of Kelvin-Helmholtz Instabilities in Stratified Ocean Flows , DANA BROWN, LOUIS GOODMAN, MEHDI RAESSI, University of Massachusetts Dartmouth — Numerical simulations of turbulence in the ocean environment are used to supplement and enhance understanding of observational data. Here, using the NGA framework (Dejardins et al., JCP 2008), direct numerical simulations (DNS) and large eddy simulations (LES) of Kelvin-Helmholtz instabilities are employed to study turbulence in presence of density stratification. Kelvin-Helmholtz instabilities have been shown to be a common source of turbulence in the ocean. Past DNS studies of Kelvin Helmholtz instabilities have compared favorably with observational data, but were limited to moderate Reynolds numbers. Here, LES is used to solve the filtered incompressible NS equations at a higher Reynolds number, Re = 10,000. The effect of increased Reynolds number on the turbulence behavior is examined in terms of velocity spectra and energy budgets.

1:52PM R7.00005 Instabilities of pancake vortices modelled by rotating ellipsoids in a stratified fluid[1] , PATRICE MEUNIER, IRPHE, Aix Marseille Univ., CNRS, Marseille, France — It is now well known that oceans contain very energetic vortices with a long lifetime. However, it is still unclear how these vortices destabilise and how much energy and mixing they can provide at different scales. We investigate here the destabilisation of an axisymmetric vortex in a stratified and non-rotating environment. The vortex is modeled by a rotating ellipsoid with various diameters and heights. The flow is visualised by shadowgraph, synthetic schlieren and Particle Image Velocimetry. Two types of instabilities have been observed, one being located on the side of the ellipsoid and the other being located at the top and bottom. The first instability is linked to the radiative instability, which is well known in the case of a rotating cylinder, and which emits internal waves with an azimuthal wave number equal to 1. The second instability generates an axisymmetric layering pattern which is reminiscent of the double diffusive instability (between angular momentum and density), observed and described theoretically in a rotating environment. This second instability might be responsible for the layering pattern found above oceanic vortices, which probably leads to a large localised mixing.

[1]ANR Grant OLA

2:05PM R7.00006 Unstable modes of a sheared pycnocline above a stratified layer , SCOTT WUNSCH, KURT KELLER, Johns Hopkins University — Internal waves incident on a sheared ocean pycnocline are studied using analytic and numerical methods. Linear analysis of the unstable modes of a sheared ocean pycnocline is used to demonstrate interactions between internal waves and shear instabilities. A new analytic solution for an asymmetric shear layer over a stratified layer is presented, illustrating modes which couple to internal waves, in addition to the well-known Holmboe modes. The robustness of these solutions is demonstrated using numerical methods for realistic shear profiles. Fully nonlinear numerical simulations illustrate the growth of these modes and demonstrate the excitation of shear instabilities by incident internal waves. The results may have implications for internal wave interactions with the ocean pycnocline and the local generation of internal solitary waves.

2:18PM R7.00007 Inertial instability of oceanic submesoscale vortices: linear analysis, marginal stability criterion and laboratory experiments , AYAH LAZAR, Geophysics and Planetary Sciences, Tel-Aviv University and Laboratoire de Meteorologie Dynamique, Ecole Polytechnique, ALEXANDRE STEGNER, Laboratoire de Meteorologie Dynamique, CNRS and Ecole Polytechnique, EYAL HEIFETZ, Geophysics and Planetary Sciences, Tel-Aviv University — Inertial instability is a possible mechanism for vertical mixing in the submesoscale ocean. The stability of axisymmetric oceanic-like vortices to inertial perturbations is investigated by linear stability analysis, taking into account the thickness and the stratification of the thermocline, as well as the vertical eddy viscosity. Numerical analysis reveals that the instability is insensitive to the vorticity profile if the intensity of the vortex is characterized by the vortex Rossby number (instead of the local normalized vorticity). This allows extending our analytical solutions for the Rankine vortex to a wide variety of oceanic cases, including results such as the analytic dispersion relation, and the marginal stability criterion. This suits oceanic conditions better than the widely used generalized Rayleigh criterion. Comparison with oceanographic data shows that our criterion permits cases that contradict the common oceanographic hypothesis for inertial instability. For instance, intense submesoscale anticyclones may be stable even with a core region of negative absolute vorticity. We corroborate our findings with large-scale laboratory experiments and find a signature of the instability on the mean-flow, which could be used in future oceanographic measurements.

2:31PM R7.00008 The Easily Excitable Baroclinic Critical Layers in Rotating, Horizontally Shearing, Vertically Stratified Flows and Their Roll-up into Vortices , PHILIP MARCUS, SUYANG PEI, CHUNG-HSIANG JIANG, PEDRAM HASSANZADEH, UC Berkeley — Baroclinic critical layers can occur in rotating, vertically-stratified, uni-directional shear flows. They are special cases of neutrally stable eigenmodes. Baroclinic critical layers have logarithmic singularities in density and vertical velocity. They differ from barotropic critical layers associated with Kelvin's cats-eyes in constant-density, uni-directional shear flows, which form at locations where the shear flow velocity matches the eigenmode's phase speed and have singularities only in stream-wise velocities. Baroclinic critical layers are excited with no special tuning of parameters by perturbations from vortices or waves. Unlike barotropic critical layers the amplitudes of baroclinic layers become large by drawing energy from the background shear. The large vertical velocities in the critical layers, coupled with the Coriolis parameter create large-amplitude vortex layers. These layers often roll-up into large coherent vortices. The baroclinic critical layers' growth and roll-up are robust: they form in cylindrical and Cartesian geometries, in Boussinesq fluids and ideal gases, and in flows with uniform and non-uniform shear and vertical stratification. However, they do not form in numerical calculations with insufficient spatial resolution or large grid dissipation.

2:44PM R7.00009 Self-Similar, Self-Replicating, Critical Layers and Vortices in Rotating, Horizontally Shearing, Vertically-Stratified Flows , SUYANG PEI, CHUNG-HSIANG JIANG, PHILIP MARCUS, PEDRAM HASSANZADEH, UC Berkeley — In a rotating,uni-directional flow with a vertical Brunt-Väisälä frequency $N(z)$ and horizontal shear σ, baroclinic critical layers (a form of neutrally stable eigenmode) form at cross-stream locations that are functions of N and the eigenmode's stream-wise wavenumber and temporal frequency. The critical layers, which are easily excited by waves or vortices, grow in amplitude, roll-up and create new vortices at the critical layers. These vortices, in turn create new critical layers. In flows with uniform σ, the process of excitation, critical layer growth, roll-up and vortex creation can self-similarly self-replicate so that the entire 3D computational domain fills with a spatially periodic lattice of large-amplitude vortices. This self-replication occurs in flows that are linearly stable (in particular, they are convectively and centrifugally stable with a uni-directional flow with no inflection points). Thus, a small, but finite-amplitude perturbation in the form of a wave or vortex fills the entire flow with large-amplitude coherent structures. This phenomenon was serendipitously discovered in calculations of linearly stable Keplerian disks and of planetary vortices in zonal flows, but also applies to large Reynolds number lab flows such as Couette flow.

2:57PM R7.00010 Turbulence, submesoscales, and the spin down of ocean fronts , JOHN TAYLOR, Department of Applied Mathematics and Theoretical Physics, University of Cambridge — Ocean fronts, regions of strong horizontal density gradients, are important for many processes in the ocean, including heat transport, CO2 uptake, water mass formation, and biological productivity. Unlike non-rotating flow, where a horizontal density gradient would lead to gravitational slumping, many ocean fronts are balanced by an along-front flow known as the "thermal wind." However, this equilibrium is unstable to a variety of instabilities, some of which generate O(1-10km) features known as submesoscales, a major focus of recent work. Despite recent progress, several important questions remain, including: How do submesoscales interact with boundary layer turbulence, and how effective are they at transferring energy to smaller scales, thereby completing a down-scale route to dissipate frontal energy. Here, we will discuss two areas of research that address these questions. First, horizontal straining by submesoscale eddies enhances horizontal density gradients. When forced by surface cooling, these regions generate localized pockets of intense turbulence. Second, wind forcing can generate time-dependent currents that trigger "symmetric instability," which is efficient at extracting frontal kinetic energy and enhances small-scale turbulent dissipation.

Tuesday, November 20, 2012 1:00PM - 3:10PM –

Session R8 Drops XII 25A - Chair: Arne Pearlstein, University of Illinois at Urbana-Champaign

1:00PM R8.00001 Internal flow and deformation of a liquid CO_2 drop rising through water[1] , LOUIS L. STEYTLER, ARNE J. PEARLSTEIN, University of Illinois at Urbana-Champaign — We report computations of the steady axisymmetric flow in and around a deformable liquid drop of CO_2 ascending through a water column under the action of buoyancy, a problem relevant to risk assessment for sub-seabed carbon sequestration and storage. In these initial computations, we consider several drop densities, corresponding to different depths in the ocean, and neglect dissolution of CO_2 into the surrounding water and formation of a hydrate film at the drop/water interface. The results, which extend our previous work (Bozzi et al., *J. Fluid Mech.*, **336**, 1-32, 1997) to the case in which the dynamic viscosities of the dispersed and continuous phases are unequal, show that the degree of deformation and internal circulation depend strongly on drop size.

[1]Supported by the International Institute for Carbon-Neutral Energy Research, sponsored by the Japanese Ministry of Education, Culture, Sports, Science and Technology.

1:13PM R8.00002 3D Droplet velocities and sizes in the Ranque-Hilsch vortex tube[1] , R. LIEW, J.C.H. ZEEGERS, Department of Applied Physics, Eindhoven University of Technology, J.G.M. KUERTEN, Faculty EEMCS, University of Twente, W.R. MICHALEK, Department of Mechanical Engineering, Eindhoven University of Technology — The Ranque-Hilsch vortex tube is a known device that is used to generate spot cooling. In this study, we experimentally investigate the behavior of small water droplets in the vortex tube by means of Phase Doppler Particle Analysis. In an experimental vortex tube, droplets were injected together with a carrier gas to form a fast rotating (up to 80.000 rpm) droplet-gas mixture. Droplet sizes, 3D velocity components, and turbulent properties were measured, showing high intensity isotropic turbulence in the core region. To investigate the cause of the high intensity turbulence, a frequency analysis was applied on the measured velocity. The frequency spectrum of the velocity is presented and indicates that wobbling of the vortex axis is the cause of the high turbulence intensity. It was expected that larger droplets have a higher radial velocity because of the larger centrifugal force. Results show, however, that small and lager droplets behave similar.

[1]This research is supported by the Dutch Technology Foundation STW, which is the applied science division of NWO, and the Technology Programme of the Ministry of Economic Affairs.

1:26PM R8.00003 The Polarization of a Diffuse Soft Particle Subjected to an Alternating Current Field , SEBASTIAN UPPAPALLI, HUI ZHAO, University of Nevada Las Vegas — The polarization of a diffuse soft particle submerged in an aqueous electrolyte and subjected to a uniform, alternating electric field is theoretically analyzed with the standard electrokinetic model. The particle consists of a rigid uncharged core and a charged diffuse polyelectrolytic shell (soft layer), permeable to ions and solvent. Our focus is on the impact of the characteristics of the soft layer including Donnan potential, the soft layer thickness and the friction coefficient of the soft layer on the dipole coefficient, characterizing the strength of the polarization. Under the limits of thin double layers and thin polyelectrolytic shells, approximate, analytical expressions to evaluate dipole moment coefficients are derived, respectively, for high-frequency and low-frequency ranges. The analytical results are compared and agree favorably with those numerically computed by the standard model. Interestingly, we discover that when the double layer is comparable to the soft layer, the dipole moment behaves qualitatively differently at different Donnan potentials. When the Donnan potential is small, the dipole moment decreases as the double layer increases. In contrast, at large Donnan potentials, the dipole moment increases with the increase of the double layer. The distinct responses to Donnan potentials are attributed to the impact of the associated double layer on the charge distribution of mobile ions inside the soft layer. The theoretical model provides a fundamental basis for interpreting the polarization of heterogeneous systems including environmental or biological colloids or microgel particles.

1:39PM R8.00004 Preventing droplet deformation during dielectrophoretic centering of a compound emulsion droplet[1] , GREG RANDALL, BRENT BLUE, General Atomics — Compound droplets, or droplets-within-droplets, are traditionally key components in applications ranging from drug delivery to the food industry. Presently, millimeter-sized compound droplets are precursors for shell targets in inertial fusion energy work. However, a key constraint in target fabrication is a uniform shell wall thickness, which in turn requires a centered core droplet in the compound droplet precursor. Previously, Bei et al. (2009, 2010) have shown that compound droplets could be centered in a static fluid using an electric field of 0.7 kV/cm at 20 MHz. Randall et al. (2012) developed a process to center the core of a moving compound droplet, though the $\sim$kV/cm field induced small ($< 5\%$) but undesirable droplet stretching. This work shows that by using macromolecular emulsifiers to strengthen the droplet's interfaces, (proteins, tunable peptides, or biotinylated streptavidin) droplet stretching can be greatly inhibited. Proof-of-principle experiments are performed in either a stagnant density-matched aquarium or a vertical channel of buoyancy-driven droplets in a $\sim$kV/cm electric field. A scaling analysis is given from a fluid mechanics and interfacial rheology perspective and we discuss the effective interfacial charge from an emulsifier and its impact on centering.

[1]Work funded by General Atomics Internal R&D.

1:52PM R8.00005 Pilot-wave dynamics in confined geometries , DANIEL M. HARRIS, JOHN W.M. BUSH, MIT — Yves Couder and coworkers have demonstrated that millimetric droplets can propel themselves along the surface of a vibrating fluid bath by virtue of their pilot-wave dynamics, and that these walking droplets exhibit several features reminiscent of microscopic quantum particles. We here present the results of an experimental investigation of droplets walking in confined geometries, giving particular attention to elucidating the dynamics and statistics of the walking droplets. The behaviour depends critically on the amplitude of the vibrational forcing, specifically, the proximity to the Faraday threshold, which determines the spatio-temporal extent of the guiding wave field. Near the Faraday threshold, we demonstrate that a coherent statistical behavior emerges from the complex underlying nonlinear dynamics, and that, as in quantum mechanics, the statistics can be readily described with a linear wave equation.

2:05PM R8.00006 Viscosity Measurement via Drop Coalescence: A Space Station Experiment , BASIL ANTAR, Retired — The concept of using low gravity experimental data together with CFD simulations for measuring the viscosity of highly viscous liquids was recently validated on onboard the International Space Station (ISS). A series of microgravity tests were conducted for this purpose on the ISS in July, 2004 and in May of 2005. In these experiments two liquid drops were brought manually together until they touched and were allowed to coalesce under the action of the capillary force alone. The coalescence process was recorded photographically from which the contact radius speed of the merging drops was measured. The liquid viscosity was determined by fitting the measured data with accurate numerical simulation of the coalescence process. Several liquids were tested and for each liquid several drop diameters were employed. Experimental and numerical results will be presented in which the viscosity of several highly viscous liquids were determined using this technique.

2:18PM R8.00007 Frictional effects on liquid marbles[1] , GUILLAUME LAGUBEAU, ANTONELLA RESCAGLIO, FRANCISCO MELO, Universidad de Santiago de Chile, CHRISTOPHE CLANET, Ladhyx, France, DAVID QUÉRÉ, PMMH-ESPCI, France — Liquid marbles are liquid drops coated with hydrophobic grains. This kind of interfaces has a dual nature as it mixes the capillar behavior of the liquid with the granular properties of the grains cap. This induces remarkable properties that have been shown recently and used for example for stabilization of liquid foams. Nevertheless, their fundamental mechanisms have yet to be fully understood. We present here an experimental study of statics and dynamics effects of the solid friction between grains on the liquid marbles properties.

[1]This study is financed by Fondecyt postdoctoral award 3120120.

2:31PM R8.00008 Holy Balls!: Part Deux , JESSE BELDEN, MICHAEL JANDRON, Naval Undersea Warfare Center, TADD TRUSCOTT, Brigham Young University — A Waboba® (WAter BOuncing BAll) demonstrates remarkable water skipping behavior, even at relatively large impact angles. The highly compliant nature of these elastic spheres results in significant deformation into a disk-like shape upon impact. The increased wetted area and force coefficient generates a large hydrodynamic force that more readily lifts the ball off the water surface. However, elasticity introduces some surprising phenomena, such as material waves that propagate on the sphere and interact with the water cavity. Depending upon impact conditions, material waves may propagate in various directions combining to create multiple modes of deformation and complicated fluid-structure interactions. Furthermore, the timescales of deformation and wave propagation depend on the material properties and impact conditions. In this talk, we will discuss skipping regimes in terms of impact parameters and material properties and relate failed skipping behavior to the structure-fluid interaction caused by deformation. The critical timescales for deformation, wave propagation and collision will be related to the relevant physical parameters of the problem.

2:44PM R8.00009 The effects of the impact velocity of water droplets on the surface features in a rain field[1] , REN LIU, XINAN LIU, JAMES DUNCAN, University of Maryland, College Park — The characteristics of the shape of a water surface in response to the impact of simulated raindrops are studied experimentally. The experiment is carried out in a 1.22-m by 1.22-m water pool with a water depth of 0.3 m. Simulated raindrops are generated by an array of 22-gauge needles that are attached to a flat plate and connected to a water reservoir with soft hoses. A random translational motion is added to the plate to produce a randomness of the drop impact locations for each needle. The height of the flat plate relative to the water surface of the pool is varied from 1 m to 3 m to vary the impact velocities of the water drops. The surface features due to the impact of the drops, including the crown, stalk and ring waves are measured with a cinematic laser-induced- fluorescence (LIF) technique. It is found that the size of the crown, the height of the stalks and the wavelength of the waves vary with the impact velocity of drops. A comparison of these features with those found in a single drop impact is highlighted.

[1]This work is supported by the National Science Foundation, Division of Ocean Sciences under grant OCE962107.

2:57PM R8.00010 ABSTRACT WITHDRAWN –

Tuesday, November 20, 2012 1:00PM - 3:10PM –

Session R9 Magnetohydrodynamics 25B - Chair: Kai Schneider, Aix-Marseille University

1:00PM R9.00001 ABSTRACT WITHDRAWN –

1:13PM R9.00002 Turbulent transfer and secondary flow patterns in transitional MHD duct flows under the non-uniform magnetic field , HIROMICHI KOBAYASHI, Keio University, YOSHIHIRO OKUNO, Tokyo Institute of Technology — Large-eddy simulation (LES) of transitional turbulent duct flows is carried out in a liquid metal MHD power generator, and the influence of the non-uniform magnetic field on the turbulent flows is examined. As increasing the magnetic flux density (or Hartmann number), the turbulence is suppressed downstream of electrodes. The higher Hartmann number modulates the mean velocity profile to the M-shaped velocity one (the so-called sidewall jet) in the plane parallel to the external magnetic field. The velocity profiles are modulated more strongly with the magnetic flux density. In the higher Hartmann number, the wall-shear stress in the sidewall layer becomes large and the sidewall jets transit to turbulence. The sidewall jets in the MHD turbulent duct flows have a similar mean velocity profile of the non-MHD wall jets with outer scaling as well as the profiles of Reynolds shear stress with the opposite sign and two maxima for the turbulent intensities in a sidewall jet. The Lorentz force suppresses the vortices of the secondary mean flow near the Hartmann layer for low Hartmann number, whereas the secondary vortices remain near the Hartmann layer for high Hartmann number.

1:26PM R9.00003 Experimental Investigation on Liquid Metal Flow Distribution in Insulating Manifold under Uniform Magnetic Field , MASATO MIURA, YOSHITAKA UEKI, TAKEHIKO YOKOMINE, TOMOAKI KUNUGI, Department of Nuclear Engineering, Kyoto University — Magnetohydrodynamics (MHD) problem which is caused by interaction between electrical conducting fluid flow and the magnetic field is one of the biggest problem in the liquid metal blanket of the fusion reactor. In the liquid metal blanket concept, it is necessary to distribute liquid metal flows uniformly in the manifold because imbalance of flow rates should affect the heat transfer performance directly, which leads to safety problem. While the manifold is insulated electrically as well as the flow duct, the 3D-MHD effect on the flowing liquid metal in the manifold is more apparent than that in straight duct. With reference to the flow distribution in this concept, the liquid metal flow in the electrical insulating manifold under the uniform transverse magnetic field is investigated experimentally. In this study, GaInSn is selected as working fluid. The experimental system includes the electrical magnet and the manifold test section which is made of acrylic resin for perfectly electrical insulation. The liquid metal flows in a non-symmetric 180°-turn with manifold, which consists of one upward channel and two downward channels. The flow rates in each channel are measured by electromagnetic flow meters for several combinations Reynolds number and Hartman number. The effects of magnetic field on the uniformity of flow distribution are cleared.

1:39PM R9.00004 Numerical investigation of the turbulent MHD flow in a circular pipe with transverse magnetic field[1] , XAVIER DECHAMPS, Department of Aero-Thermo-Mechanics, Université Libre de Bruxelles, MICHEL RASQUIN, Argonne National Laboratory and University of Colorado Boulder, GÉRARD DEGREZ, Department of Aero-Thermo-Mechanics, Université Libre de Bruxelles — In modern industrial metallurgical processes, external magnetic fields are often applied to control the motion of liquid metals by a non-intrusive means. The desired results are for example the damping of unwanted motions or the homogenization of a liquid zone in a partially solidified ingot. Because of the commonly appearing parameters in these processes, one can assume the quasi-static assumption for the magnetohydrodynamic equations. Here we are interested in the numerical study of the turbulent flow of a liquid metal inside an electrically insulated pipe with a transverse uniform magnetic field. For this purpose, we will use a hybrid spectral/finite element solver, which allows to study complex flows in Cartesian and axisymmetric geometries. For the case of interest, we consider a bulk Reynolds number of 8200 and a Hartmann number ranging between 5 and 30. Here, the main points of interest are the evolution of the skin friction coefficient as a function of the ratio of the Hartmann number Ha over the Reynolds number Re (with $0 < \mathrm{Ha/Re} < 75\mathrm{x}10^{-4}$) as well as the energy budget (viscous, Joule and numerical dissipations, kinetic energy production) in a cross-section. These results will determine the transition point between laminar and turbulent flows.

[1]Support of the Fonds Nationals de la Recherche Scientifique (FNRS) is aknowledged.

1:52PM R9.00005 Instabilities in Turbulent Magnetized Spherical Couette Flow , MATTHEW M. ADAMS, DANIEL S. ZIMMERMAN, University of Maryland, College Park, SANTIAGO TRIANA, None, DANIEL P. LATHROP, University of Maryland, College Park — We present experimental studies of the turbulent shear flow of a conducting fluid in a spherical-Couette device in the presence of a magnetic field. Our experimental apparatus consists of an outer spherical shell concentric with an inner sphere, which both rotate independently. The geometry of the experiment makes these studies applicable to geophysical and astrophysical bodies. Liquid sodium fills the gap between the inner sphere and the shell. We apply an axial magnetic field of varying strength to study the influence of the applied field on the dynamics of the flow. 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, and take direct fluid pressure measurements. We use these to study instabilities that appear as the applied field is increased, for the case of a stationary outer sphere, and for both spheres rotating independently, and compare with theory and numerical predictions.

2:05PM R9.00006 Numerical Study for the MHD Homogeneous Decaying Turbulence under the System Rotation , MASAYOSHI OKAMOTO, DAIYU NAKAJIMA, Shizuoka University — In this study the MHD homogeneous decaying turbulence under the system rotation is directly simulated by means of the pseudo-spectral method. The decaying rates of the kinetic and magnetic energy are suppressed due to the rotation effect like that of the HD turbulence. The small-scale anisotropy of the velocity and magnetic fields is reversed in comparison with the weak anisotropy of the Reynolds and Maxwell stresses. The weak transformation of the energy and stress from the velocity field to the magnetic one occurs under the system rotation, but in the small-scale region the magnetic energy is converted into the kinetic energy. In the strong rotation case, the slope of the energy spectrum is steeper than Kolmogorov one. In the strong rotation case, the vortex structures are aligned with the rotation axis.

2:18PM R9.00007 Coherent vorticity and current density simulation of three-dimensional magnetohydrodynamic turbulence using orthogonal wavelets , KATSUNORI YOSHIMATSU, NAOYA OKAMOTO, YASUHIRO KAWAHARA, Nagoya University, KAI SCHNEIDER, Aix-Marseille Universite, MARIE FARGE, Ecole Normale Superieure — A simulation method to track the time evolution of coherent vorticity and current density, called coherent vorticity and current density simulation (CVCS), is developed for three-dimensional (3D) incompressible magnetohydrodynamic (MHD) turbulence. The vorticity and current density fields are, respectively, decomposed at each time step into two orthogonal components, the coherent and incoherent fields, using an orthogonal wavelet representation. Each of the coherent fields is reconstructed from the wavelet coefficients whose modulus is larger than a threshold, while their incoherent counterparts are obtained from the remaining coefficients. The two threshold values depend on the instantaneous kinetic and magnetic enstrophies. In order to compute the flow evolution, one should retain not only the coherent wavelet coefficients but also their neighbors in wavelet space, and the set of those additional coefficients is called the safety zone. CVCS is performed for 3D forced incompressible homogeneous MHD turbulence without mean magnetic field for a magnetic Prandtl number equal to unity and with 256^3 grid points. The quality of CVCS is assessed by comparing the results with a direct numerical simulation. It is found that CVCS with the safety zone well preserves the statistical pred

2:31PM R9.00008 Intrinsic rotation of toroidally confined magnetohydrodynamics , JORGE MORALES, WOUTER J.T. BOS, LMFA-CNRS, Ecole Centrale de Lyon - Universite de Lyon, Ecully, France, KAI SCHNEIDER, M2P2-CNRS & CMI, Aix-Marseille University, Marseille, France, DAVID C. MONTGOMERY, Department of Physics and Astronomy, Dartmouth College, Hanover, New Hampshire, USA — Time-dependent three-dimensional toroidal visco-resistive MHD pseudo-spectral computations are performed, using the recently developed penalization method for enforcing the boundary conditions. An imposed toroidal magnetic field is present and the current is driven by an imposed toroidal electric field. Both poloidal and toroidal rotation result, and depend strongly on the shape of the toroidal cross section and the value of the Hartmann number. Net toroidal rotation results from a departure from up/down symmetry in the cross-sectional boundary shape. By increasing the Hartmann number, the plasma seeks out a characteristic configuration in which the velocity aligns approximately with the magnetic field lines. The resulting flow is characterized by both toroidal and poloidal rotation, starting from initial conditions in which such flows are absent. Ideal MHD equilibrium considerations appear not to play an important role.

2:44PM R9.00009 Experimental investigation of heat transfer in free-surface MHD flow[1] , J. RHOADS, Princeton University, A. KATZENSTEIN, Eckerd College, E. EDLUND, P. SLOBODA, E. SPENCE, H. JI, PPPL — The presence of a strong external magnetic field can significantly alter the dynamics of large and small scale features within the flow. In particular, turbulent eddies with vorticity non-parallel to the magnetic field are strongly damped. This anisotropization of the turbulence may be critically important for heat transport in flowing liquid metal walls in a fusion reactor. Experiments have been conducted in the Liquid Metal Experiment (LMX) using a GaInSn eutectic alloy as a working fluid to investigate these effects. These experiments considered free-surface, wide aspect-ratio flows up to 20 cm/s through a channel situated in a magnetic field up to 2 kG, corresponding to a Reynolds number up to $\mathrm{Re} \approx 10^4$ and a Hartmann number up to $\mathrm{Ha} \approx 50$. Resistive heaters were placed on the free surface and the fluid temperature downstream was monitored by an array of thermocouples and an infrared camera. The relationship between Nusselt number and Hartmann number will be presented.

[1]Work supported under contract DE-AC02-09CH11466.

2:57PM R9.00010 Falling-Ball Rheometry of Magnetized Ferrofluids in the Presence of Magnetic Particle Threads[1] , A.F. CALI, A.D. TRUBATCH, P.A. YECKO, Montclair State University — To probe the rheology of magnetized ferrofluid on a millimeter scale, we examine the paths of glass spheres (500 micrometer diameter) falling in a bulk ferrofluid under a uniform magnetic field, applied at a fixed angle with respect to the horizontal. Visualization of the ball trajectories is accomplished by the capture of high-resolution, X-ray phase-contrast images on high-speed, digital video. The drag on each sphere is determined by measurement of its terminal velocity. The effective viscosity of the ferrofluid is observed to be anisotropic: the terminal velocity of a sphere depends on the angle of the applied magnetic field. Specifically, the drag is greater when the applied field is normal to the vertical path of the falling sphere than when parallel. We propose that this effect is due the field-induced formation of magnetic particle "threads" having diameter of a few microns (∼1000 nanoparticles) and lengths that span the container (a few millimeters). These threads form a dense parallel array aligned with the applied magnetic field and are visible in phase-contrast images of the bulk ferrofluid. We model the drag experienced by the falling ball as originating from the falling-ball Stokes flow through a fixed array of rigid threads with an inter-penetrating fluid.

[1]We acknowledge support by N.S.F. Grant No. 1016383

Tuesday, November 20, 2012 1:00PM - 2:57PM –

Session R10 General Instability II 25C - Chair: Anne Cros, Universidad de Guadalajara

1:00PM R10.00001 Elasto-Capillary Coalescence of Multiple Parallel Sheets , AMIR GAT, MORTEZA GHARIB, California Institute of Technology — We analyzed two-dimensional clamped parallel elastic sheets which are partially immersed in liquid as a model for elasto-capillary coalescence. The existing literature studied this problem via minimal energy analysis of capillary and elastic energies of the post-coalescence state. Utilizing modal stability analysis and asymptotic analysis, we studied the stability of the configuration before the coalescence occurred. Our analysis revealed previously unreported relations between viscous forces, body forces, and the instability yielding the coalescence. A mathematical description of the process creating the hierarchical coalescence structure was obtained and yielded that the mean number of sheets per coalesced region is limited to the subset 2^N where N is the set of natural numbers. Our results were compared with experimental data and a reasonable agreement was observed.

1:13PM R10.00002 Influence of gravity on flutter of cantilevered pipes conveying fluid[1] , JAVIER RIVERO, MIGUEL PEREZ-SABORID[2], Dep. Ingeniería Aeroespacial y Mecánica de Fluidos, Universidad de Sevilla, Sevilla 41092, Spain — We have considered the dynamics of the nonlinear interaction between a flexible pipe and the conveyed fluid in the presence of gravity. The stability of the system (fllutter and buckling) depends on parameters such as the dimensionless fluid flow rate, the gravity to bending stiffness ratio and the fluid to pipe mass ratio and it has been studied in detail both numerically and experimentally. It has also been found that the stabilizing or destabilizing effects of fluid flow depends crucially on the direction of gravity respect to the undeformed midline of the pipe. We have also computed the post-critical behavior of the system by solving the full nonlinear equations of the problem and analyzed the transfer of energy within the system in the nonlinear regime. We have formulated the problem in terms of the angles of the midline of the pipe instead of its transverse displacements, so that we can deal with large deflections without recurring to the quasi-linear approximations concerning the pipe curvature usually made in the literature.

[1]Supported by the Ministerio de Educación, Cultura y Deportes of Spain under grant DPI 2010-20450 C03-02.
[2]Author to whom correspondence should be addressed

1:26PM R10.00003 ABSTRACT WITHDRAWN –

1:39PM R10.00004 Sky dancer: the threshold of the buckling instability , ANNE CROS, RODOLFO IBARRA NUÑO, Dpto de Fisica, Universidad de Guadalajara, Mexico, BASTIEN MICHON, IUT GTE, Universite Aix-Marseille III, France — The sky dancer is a fabric-made large vertical tube which may "dance" above an air blower. We reproduced in laboratory reduced models, with diameters between 2.2 and 3 cm, and lengths varying from 45 to 95 cm. We measured the air speed and the pressure at the tube basis for the lower threshold of the tube. This threshold separates two regimes: the first one which appears for the lower velocities, when the tube falls down because of its own weight, from the second one, which corresponds to the stable regime, for which the tube stands up. Our measurements show that the air pressure thresholds get values equal to the pressure performed by the tube weight. We discuss our results and compare them with previous studies which dealt with thicker-walled tubes.

1:52PM R10.00005 Study of Waving of grass using soap film , RAVI SINGH, SHREYAS MANDRE, AMALA MAHADEVAN, Brown University, L. MAHADEVAN, Harvard University, MAHESH BANDI, Okinawa Institute of Science and Technology — Wind blowing over a grass field incites synchronized response from the grass blades, which appear as waves. This effect is called Mo-nami in a terrestrial setting, while in an aquatic setting it is termed as Ho-nami. We use a combination of experimental observations, numerical simulations and theoretical analysis to understand this effect. The experiment is conducted in two-dimensional realization of these phenomenon in a gravity driven soap film tunnel. Nylon filaments attached to the boundaries of the soap film play the role of the grass. We provide a preliminary characterization of this analog model for synchronized oscillations of grass.

2:05PM R10.00006 ABSTRACT WITHDRAWN –

2:18PM R10.00007 Vortex shedding experiments with Ekman friction[1] , PAUL W. FONTANA, ELIZABETH BRASSEALE, JOHN P. GOERTZ, DAVID RASCHKO, JEMIN SHIM, Seattle University — We present an experimental investigation of the Strouhal instability in flowing soap films. In vortex shedding by a cylinder, Roshko's formula for the one-to-one relationship between the Strouhal number $St = fD/U_\infty$ and the Reynolds number $Re = U_\infty D/\nu$ is well-established empirically and in numerical simulations. (Here f is the vortex shedding frequency, D is the cylinder diameter, U_∞ is the upstreem flow speed, and ν is the kinematic viscosity.) However, the effect of homogeneous linear drag (Ekman friction) on the relationship has not previously been investigated. It has generally been assumed to be unimportant, but it is plausibly hypothesized to decrease the shedding frequency. If so, then ignoring drag may lead to inaccurate estimates of Re based on St in situations where Ekman friction is significant. We address the question with quasi-two-dimensional vortex shedding experiments in soap films in which we are able to resolve viscosity and surface friction. The results shed light on the instability mechanism of vortex shedding in particular, and they also demonstrate how precise measurement of both viscosity and surface friction is necessary to make soap film experiments useful for quantitative investigations of stability more generally.

[1]Supported by the National Science Foundation under Grant No. CBET-0854509.

2:31PM R10.00008 On the dynamics of gaseous detonation in porous inert media , ROMAN SEMENKO, ASLAN KASIMOV, King Abdullah University of Science and Technology, BORIS ERMOLAEV, Semenov Institute of Chemical Physics — We consider a one-dimensional detonation wave propagating through a mixture of detonable gas in a porous inert medium. We assume the presence of solid metal spheres in a tube which incur the losses of the momentum and heat. The main goal of this work is to understand the role played by the losses in the structure of the steady state solutions and to analyze their linear stability.

2:44PM R10.00009 Unstable Homogeneous Boiling , SAI RAMACHANDRAN, ASGHAR ESMAEELI, Southern Illinois University Carbondale — Direct Numerical Simulations are performed to study homogeneous nucleation of vapor nuclei in a pool of highly heated liquid. The process is highly unstable and leads to wrinkling at the surface of the bubbles and evaporative mass fluxes that are several order of magnitude larger than those of stable boiling. The goal is to gain fundamental understanding about the phenomenon by comparing the various growth rate of the nuclei against that predicted by classical theory and also shed some lights on the nature of the surface instability by comparing the growth rate of the protrusions with those predicted by linear stability theories for spherical systems and Landau's theory for instability of laminar flames.

Tuesday, November 20, 2012 1:00PM - 3:10PM –

Session R11 Microfluidics: Particles 26A - Chair: Alexander Alexeev, Georgia Institute of Technology

1:00PM R11.00001 Three dimensional open cavity flow for the continuous separation of suspended particles , JORGE A. BERNATE, Stanford University, COLIN PAUL, Johns Hopkins University, CHENGXUN LIU, LIESBET LAGAE, IMEC, KONSTANTINOS KONSTANTOPOULOS, ZACHARY GAGNON, GERMAN DRAZER, Johns Hopkins University — We present a microfluidic platform for the continuous separation of suspended particles based on their size and settling velocity, which relies on the reorientation of the flow field created by applying a pressure gradient across and along a periodic array of open cavities. The flow along the open cavities deflects different particles to a different degree depending on the extent to which they penetrate into the open cavities. Two regimes can be distinguished depending on the ratio r between the settling velocity of the particles and their velocity across the cavities. When $r \approx 1$, heavier particles settle deeper into the open cavities and deflect more than lighter ones. When $r \ll 1$, smaller particles are advected deeper into the cavities by the flow and deflect more than larger ones. We probe these regimes by separating spherical particles of different size and density at different flow rates. We show the potential of this platform to be used as a microfluidic *centrifuge* depleting RBCs and enriching spiked MCF-7 cancer cells. This platform can be easily integrated with external fields resulting in a potentially versatile technique. In particular, we use dielectrophoretic forces for the high-throughput separation of particles of the same size.

1:13PM R11.00002 Microfluidic separation of motile sperm with millilitre-scale sample capacity , REZA NOSRATI, MARION VOLLMER, LISE EAMER, University of Toronto, KRISTA ZEIDAN, MARIA C. SAN GABRIEL, ARMAND ZINI, McGill University, DAVID SINTON, University of Toronto — Isolating motile from non-motile spermatozoa has been a challenge since the establishment of *in vitro* fertilization. Microfluidic approaches have been employed for this purpose, but current devices are limited by low sample volume. Here, we present a high-throughput microfluidic device that separates spermatozoa from one millilitre of raw semen sample based on the hydrodynamic characteristics of swimming sperm in a confined geometry. The device consists of two layers: an outer injection ring on top aligned with a network of radial microchannels at the bottom guiding motile sperm into an inner collection chamber. This approach (1) maximizes exposure of the sperm to the fluid channels, (2) maximizes surface area density (3) prevents fluid flow bias, and (4) employs a non-Newtonian viscoelastic medium consistent with the *in vivo* environment. Tests with human and bull spermatozoa indicate an increase in motile sperm concentration from 62.2% in raw semen to 99.2% in separated sample combined with a higher incidence of normal morphology. DNA integrity testing is currently underway. In conclusion, we present an effective one-step procedure to perform semen purification and separation on a millilitre-scale with clinically relevant numbers.

1:26PM R11.00003 Single-stream inertial focusing of microparticles across laminar streamlines through geometry-induced secondary flows , ARAM CHUNG, DIANNE PULIDO, JUSTIN OKA, MAHDOKHT MASAELI, HAMED AMINI, DINO DI CARLO, Department of Bioengineering, UCLA — The ability to continuously control microparticle position in a confined microchannel is remarkably useful for a wide range of biomedical studies Current state-of-the-art systems to achieve particle focusing require either complex external setups accompanying complicated fabrication steps or logistically burdensome sheath fluid. Using the fluid inertia acting on particles in microchannels has been introduced to address these limitations since inertia can position particles precisely in a predictable manner. Previous work has predominantly demonstrated multiple focusing streams however here we present a novel method that initially randomly distributed microparticles can be focused into a single-stream by (1) introducing a series of cylindrical pillars in a microchannel or (2) locally modifying channel geometry Briefly, the combination of inertial focusing upstream and a pair of local helical secondary flows induced by the obstacles or steps in channel height allows for migration of microparticles to a single position in a high-throughput manner We present comprehensive numerical and experimental studies and results of the particle-fluid interaction and focusing mechanism, characterize the role of flow deformation, determine focusing accuracy and discuss potential applications.

1:39PM R11.00004 Size-based dielectrophoretic particle sorting in a microfluidic device with thermal effects , BARUKYAH SHAPARENKO, University of Pennsylvania, HAN-SHENG CHUANG, National Cheng Kung University, HOWARD HU, HAIM BAU, University of Pennsylvania — Dielectric particles in a dielectric medium experience a force known as dielectrophoresis (DEP) when subjected to a nonuniform electric field. Since this DEP force is proportional to the particle volume, it is well-suited for size-based particle sorting. Additionally, we use the geometric constraints of a narrow segment of microchannel to aid sorting by means of pinched flow fractionation (PFF). We found that in microfluidic devices, thermal effects due to Joule heating may be important. Temperature variations of the electrical material properties give rise to additional forces, including electrothermal flow in the medium and a thermal DEP force acting on the particles. We combine PFF and DEP in series to create a microfluidic sorting device, with an interdigitated array of five L-shaped electrodes and five outlet channels permitting the sorting of up to five different particle sizes. The sorting process is regulated by on-chip voltage control, so that the same device is easily adaptable to different particle sizes. We compare computed particle trajectories from an analytical solution with experimental sorting results.

1:52PM R11.00005 Particle collision dynamics in periodic asymmetric microfluidic obstacle arrays for rare cell capture , JAMES SMITH, JASON GLEGHORN[1], BRIAN KIRBY, Sibley School of Mechanical and Aerospace Engineering, Cornell University — Particle–obstacle collision dynamics in periodic microfluidic obstacle arrays are presented in the context of microfluidic devices for the capture of rare cells, such as circulating tumor cells (CTCs). A coupled CFD–particle advection simulation was used to calculate particle trajectories for low Reynolds number, low Stokes number flows. A rich range of deterministic transport modes was identified as a function of array geometry, and the resulting particle size-dependent collision rate highlights the usefulness of these arrays for high-efficiency, high-purity rare cell capture. A reduced-order model, assuming unidirectional flow and infinitesimal obstacles, captures most of the details of transport in these systems with an $O(10^4)$ computational saving; this model is a useful tool for rapidly exploring a large design space and optimizing geometries for a specific rare cell capture application. Results of the CFD simulations, reduced-order ballistic models, and experiments with polystyrene particles and cancer cells indicate that array geometry is central to rare cell capture and that simple models can be used to inform the design of these microfluidic devices.

[1]Present affiliation: Department of Chemical and Biological Engineering, Princeton University

2:05PM R11.00006 Size based separation of micro-particles using adhesive ciliated surfaces , ANURAG TRIPATHI, Dept. of Chemical and Petroleum Engineering, University of Pittsburgh, Pittsburgh, PA, 15261, AMITABH BHATTACHARYA, Dept. of Mechanical Engineering, Indian Institute of Technology Bombay, Powai, Mumbai, India, ANNA BALAZS, Dept. of Chemical and Petroleum Engineering, University of Pittsburgh, Pittsburgh, PA, 15261 — Separation of different size micro-particles in microfluidic devices is important for many biomedical applications. Various techniques ranging from active dielectrophoresis to passive separation using the concept of inertial microfluidics have been used previously. We propose a novel separation mechanism of micro-particles using active cilia arrays with adhesive tips. A near complete separation of micro-particles can be achieved for low Reynolds number (Re$\sim$0.1) regime where separation mechanisms based on inertial effects will be of little use. By means of Lattice Boltzmann simulations, we show that mixture of two different size particles with size ratio greater than or equal to two can be nearly completely separated by tuning adhesion strength, actuation frequency and cilia stiffness.

2:18PM R11.00007 Size-dependent cell separation and enrichment using double spiral microchannels , GUOQING HU, CHAO LIU, LNM, Institute of Mechanics, Chinese Academy of Sciences, JIASHU SUN, XINGYU JIANG, National Center for NanoScience and Technology, Beijing — Much attention has been directed toward microfluidic technologies that can help improve circulating tumor cells (CTCs) separation from the blood sample. In the present work, we develop a double spiral microfluidic platform with one inlet and three outlets that allows for passive, label-free tumor cell enrichment with high throughput and efficiency, inspired by the single spiral cell sorter. The curved channel induces a Dean drag force acting on cells to compete with the inertial lift, resulting in large tumor cells to be focused and deflected into the middle outlet while small hematologic cells are removed from the inner outlet. We continuously isolated and enriched the rare tumor cells (MCF-7 and Hela cells) from diluted whole blood using the same geometry. At a spike ratio of 100 tumor cells per million hematologic cells, 92.28% of blood cells and 96.77% of tumor cells were collected at the inner and middle outlet, respectively, at the throughput of 33.3 million cells per minute. A numerical model is developed to simulate the Dean flows inside the curved geometry and to track the particle/cell trajectories, which is validated against the experimental observations and serves as a theoretical foundation in optimizing the operating conditions.

2:31PM R11.00008 Thin films with self-assembled monolayers embedded on their surfaces , MD. SHAHADAT HOSSAIN, BHAVIN DALAL, SATHISHKUMAR GURUPATHAM, IAN S. FISCHER, New Jersey Institute of Technology, NADINE AUBRY, Carnegie Mellon University, PUSHPENDRA SINGH, New Jersey Institute of Technology — We have recently shown that the capillarity-based process for self-assembling particle monolayers on fluid-liquid interfaces can be improved by applying an electric field in the direction normal to the interface. The electric field gives rise to repulsive dipole-dipole forces amongst the particles causing them to move apart, and thus allowing them to move freely without blocking one another. The latter is important in the formation of virtually defect-free monolayers with long-range order. In this talk, we present a technique for freezing these expanded monolayers onto the surface of a flexible thin film. The technique involves assembling the monolayer on the interface between a UV-curable resin and a fluid which can be air or another liquid, and then curing the resin by applying UV light. The monolayer becomes embedded on the surface of the solidified resin film.

2:44PM R11.00009 ABSTRACT WITHDRAWN –

2:57PM R11.00010 Using chiral structures to enhance particle deposition in microfluidic devices , YUNJI GU, ZACHARY MILLS, ALEXANDER ALEXEEV, Georgia Tech — We used three dimensional computer simulations to examine the deposition of nanoparticles suspended in a fluid flowing through a microchannel that encompasses a periodic array of chiral structures. The channel was filled with a viscous fluid and tracer particles were used to simulate the suspended nanoparticles. The structures induce secondary flows in the fluid, which enhance mixing and in turn induce more rapid deposition of nanoparticles on channel walls. To model the system, we employed a lattice Boltzmann model coupled with a Brownian dynamics model. To investigate how the chiral structures influence the deposition rate, we systematically varied three parameters in the system: the pitch and radius of the chirals, and the spacing between structures in the array. Our simulations revealed that structures enhance nanoparticle deposition and the effect is more pronounced at larger Peclet numbers. Furthermore, we established the optimal geometry of the structures leading to increased particle deposition on the microchannel walls. Our findings could be useful for improving microscale sensory devices.

Tuesday, November 20, 2012 1:00PM - 3:10PM –

Session R12 Vortex VIII 26B - Chair: Paul Krueger, Southern Methodist University

1:00PM R12.00001 A Fluid-solid Numerical Model for the Analysis of Bio-inspired UUV , SANTANU MITRA, Virginia Tech, NAGENDRA KRISHNAMURTHY, DANESH TAFTI, Mechanical Engineering Department, Virginia Tech, SHASHANK PRIYA, Bio-Inspired Materials and Devices Laboratory (BMDL), Centre for Energy Harvesting Materials and Systems (CEHMS), Virginia Tech, VA - 24060 — Bio-inspired Unmanned Underwater Vehicles (UUVs) have potential applications for surveillance, monitoring climate change, magnetic field pattern, and migration of species. In past few years, several underwater organisms have been utilized as a source of inspiration for developing new generation of UUVs such as Mantra Ray, Squids, Dolphins and Jellyfish. In our research, we have utilized rowing form of jellyfish as a bio-inspiration and focused our attention on understanding the propulsion mechanism of medium to large diameter ($\sim$40-50 in) species. The motion of the jellyfish results a two-way coupled fluid-solid interaction problem. We provide new types of forcing functions in our model for actuating artificial jellyfish nodes employing flexible muscles such as SMA to make it energy efficient. The present simulation method uses 2-D fluid elements in the framework of the N-S based Immersed Boundary Method (IBM) and loosely coupled plane strain hyperelastic structural elements. This study will be useful in the accurate calculation of pressure distribution on the submerged autonomous vehicle and evaluation of maximum blocking stress in order to design novel actuator systems in terms of energy efficiency. Preliminary work towards achieving these final objectives will be presented.

1:13PM R12.00002 Vortex formation analysis of a piston-cylinder apparatus with passively varying output inspired by jellyfish[1] , ALEX VILLANUEVA, SHASHANK PRIYA, Virginia Tech — The flow analysis of a robotic jellyfish (Robojelly) has led to the observation of an increase in performance due to passive flexible margin. Flexible margin are common on animals using an oscillating mode of propulsion. The understanding of flexible margins is therefore important for a better understanding of animal propulsion and bio-inspired propulsion. This work focuses on analyzing the effects of stiffness and geometry of flexible margins. A piston-cylinder apparatus was used with flexible margin at the output to test the different flexible margin configurations. These results characterize the effects of the different flexible margin parameters on vortex circulation and size.

[1]Office of Naval Research through contract number N00014-08-1-0654.

1:26PM R12.00003 Perturbation response of model vortex rings and dipoles , CLARA O'FARRELL, JOHN O. DABIRI, California Institute of Technology — Jetting swimmers, such as squid or jellyfish, propel themselves by forming axisymmetric 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. In contrast, two-dimensional vortex dipoles have been found to grow well beyond the physical limit observed in axisymmetric vortex rings. Previously, the Norbury and Pierrehumbert families of vortices have been used as models for axisymmetric vortex rings and two-dimensional dipoles respectively, and the response of these two families to shape perturbations has been characterized. In this study, we improve upon the Norbury and Pierrehumbert models, using nested contours to obtain more realistic models for experimentally-generated vortex rings and dipoles. The resulting vortices are subjected to shape perturbations akin to those previously introduced to members of the Norbury and Pierrehumbert families, and their response is characterized.

1:39PM R12.00004 Experimental Investigation of a Fluidic Oscillator for Application to Pulsed-Jet Propulsion , ANNIE VAHEDIPOUR, PAUL KRUEGER, Southern Methodist University — A fluidic oscillator with no moving parts is configured with nozzles at the exit ports and is investigated experimentally to assess its performance in a configuration appropriate for continuous pulsed-jet propulsion. Oscillation frequency was controlled via the length of an external feedback tube. Performance of the oscillator was quantified by pressure measurements throughout the device, time-averaged thrust measurements, and digital particle image velocimetry (DPIV) measurements of the jet flow. Feedback tube lengths in the range 0.4 – 2 m and two flow rates (corresponding to mean jet Reynolds numbers of 9150 and 13500) were tested. Similar to prior studies, decreasing the feedback tube length and increasing the flow rate increased the oscillation frequency. However, no backflow was observed in the non-active outlet. Irregular oscillations were observed at higher frequency, but active occlusion of the feedback tube provided on/off switching of the oscillations. DPIV measurements showed formation of vortex rings at the initiation of a jet pulse, but these did not dominate the flow as the pulse durations were long for the frequency range studied.

1:52PM R12.00005 Dynamics of a Vortex Pair Impinging on a Horizontal Ground Plane , DANIEL ASSELIN, CHARLES WILLIAMSON, Cornell University — We study the effect of a solid boundary on the dynamics and instabilities of a pair of counter-rotating vortices. An isolated vortex pair is typically subject to a short-wave elliptic instability and a long-wave Crow (1971) 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. Using LIF, our facility is able to visualize both the primary and secondary vortices separately, depending on how we introduce the fluorescent dye. The long-wave Crow instability, when interacting with the wall, can cause significant axial flow, resulting in a periodic concentration of fluid at the peaks of each wavy vortex tube and a corresponding evacuation of fluid from the troughs. We are interested to determine the cause of these axial flows and to understand the vortex dynamics leading to what appear to be rebounding vortex ring structures. The vortex dynamics leading to strong axial flows seem to be a fundamental mechanism by which coherent vortex structures, such as vortex pairs or vortex rings (Lim, 1989), break up in the presence of a wall.

2:05PM R12.00006 Impact of a vortex dipole with a semi-infinite plate , SEAN D. PETERSON, University of Waterloo, MAURIZIO PORFIRI, Polytechnic Institute of New York University — Recently, several studies have been published on small-scale energy harvesting from fluids using electro-active polymers strips. Specifically, the feasibility of harvesting energy from vortex rings via impact with a cantilevered electro-active strip has recently been demonstrated. As a first step towards developing predictive models of the energy harvesting capacity of this modality, we develop a simplified two-dimensional representation of the vortex ring-deformable structure interaction, in which the vortex ring is modeled as a Lamb dipole, and the cantilevered deformable strip is replaced with a semi-infinite rigid plate. The interaction is explored numerically for a range of dipole Reynolds numbers from 500 to 3000, based upon the convection speed and dipole radius. The initial dipole trajectory results in an impact with the semi-infinite plate at its tip. As the dipole approaches, vorticity is induced in the boundary layer along the wall, which eventually separates and joins with half of the original dipole to form a secondary dipole. This interaction is similar to that of a dipole impacting an infinite wall. The other half of the original dipole merges with vorticity shed from the tip of the plate to produce another secondary dipole. The stagnation point is shifted away from the centerline of the original dipole, which differs from the case with an infinite wall. Of particular interest for the energy harvesting is the differential pressure across the semi-infinite plate, as it relates to the energy transferred to the wall in the event of a deformable, as opposed to rigid, structure, which will be discussed as well as the general flow features.

2:18PM R12.00007 Interaction of a Vortex Ring with a Thin Porous Surface , JOHN HRYNUK, DOUG BOHL, Clarkson University — The interaction of vortex rings with thin porous screens was investigated using Molecular Tagging Velocimetry (MTV). The surface porosity, defined as the ratio of the open area to total area of the screen, was held constant at $\varphi = 65\%$ while the diameter of screen wires was varied. The three screens of varying wire diameter tested were: a fine wire ($D_{wire} = 0.0178$ cm), a medium wire ($D_{wire} = 0.104$ cm) and coarse wire ($D_{wire} = 0.204$ cm). When the vortex interacted with the fine wire screen a secondary vortex formed on the upstream face of the screen that orbited the primary vortex and then convected back up stream. The primary vortex reformed immediately downstream of the screen with significantly lower strength. For medium and large wire screens additional vorticity was generated and shed from individual wires, changing the downstream vortex behavior. Secondary vortices were observed for these larger screens but they were weaker and remained in proximity to the screen. Vortex shedding from the screen wires was observed for the medium screen which delayed the reformation of the vortex ring downstream of the screen. Shed vortex pairs, from individual wires, were observed to dominate the downstream flow for the large wire screen and no vortex ring reformation was observed. Vorticity and circulation will be used to further understand the interaction process for each of these screens.

2:31PM R12.00008 Dynamics of the collision of a vortex ring with a vertical heated wall , G. GELDERBLOM, University of Twente, C.A. PALACIOS-MORALES, R. ZENIT, F.J. SOLORIO-ORDAZ, Universidad Nacional Autonoma de Mexico — We study the dynamics of the impact of a vortex ring with a vertical heated plate (at constant temperature). Laminar vortex rings were generated 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 PIV technique. The experimental results for the isothermal case are in agreement with previous investigations reported in the literature. To avoid azimuthal instabilities, we mainly conducted experiments for $L/D_0 = 1$ (where L is the piston displacement and D_0 is the cylinder inner diameter) with different wall temperatures and vortex translation velocities. For this case, secondary vortices were not observed. Using ink visualization we observed the evolution of the vortex shape. The initial circular shape evolves into a "cat head" shape after reaching the wall. The top and bottom regions of the vortex reduce and increase their vorticity, respectively. The sides are stretched and convected. An analysis of the different mechanisms leading to this shape evolution is presented and discussed.

2:44PM R12.00009 A numerical study of vorticity-enhanced heat transfer[1] , XIAOLIN WANG, Georgia Tech, SILAS ALBEN, University of Michigan — The Glezer lab at Georgia Tech has found that vorticity produced by vibrated reeds can improve heat transfer in electronic hardware. Vortices enhance forced convection by boundary layer separation and thermal mixing in the bulk flow. In this work, we simulate the heat transfer process in a 3-dimensional plate-fin heat sink. We propose a simplified model by considering flow and temperature in a 2-D channel, and extend the model to the third dimension using a 1-D heat fin model. We simulate periodically steady-state solutions. We determine how the global Nusselt number is increased, depending on the vortices' strengths and spacings, in the parameter space of Reynolds and Peclet numbers. We find a surprising spatial oscillation of the local Nusselt number due to the vortices.

[1]Support from NSF-DMS grant 1022619 is acknowledged

2:57PM R12.00010 Dynamics of SQG Point 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 SQG point vortices, retaining the vertical velocity at first order in Rossby number. The dynamics of three SQG point vortices are determined qualitatively using a phase diagram technique. Trajectories of tracer particles are then investigated using techniques such as Poincaré sections. The effect of $O(Ro)$ corrections to horizontal velocities in the derivation of the SQG equations is also examined.

Tuesday, November 20, 2012 1:00PM - 3:10PM –

Session R13 Richtmyer-Meshkov Instability II 27A - Chair: Oleg Schilling, Lawrence Livermore National Laboratory

1:00PM R13.00001 Mixing at shocked interfaces with known perturbations[1] , ANDREW COOK, Lawrence Livermore National Laboratory, CHRIS WEBER, RICCARDO BONAZZA, University of Wisconsin, BILL CABOT, Lawrence Livermore National Laboratory — We derive a growth-rate model for the Richtmyer-Meshkov mixing layer, given arbitrary but known initial conditions. The initial growth rate is determined by the net mass flux through the center plane of the perturbed interface immediately after shock passage. The net mass flux is determined by the correlation between the post-shock density and streamwise velocity. The post-shock density field is computed from the known initial perturbations and the shock jump conditions. The streamwise velocity is computed via Biot-Savart integration of the vorticity field. The vorticity deposited by the shock is obtained from the baroclinic torque with an impulsive acceleration. Using the initial growth rate and characteristic perturbation wavelength as scaling factors, the model collapses growth rates over a broad range of Mach numbers, Atwood numbers and perturbation types. The mixing layer at late times exhibits a power-law growth with an average exponent of theta=0.23.

[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. Additional support was provided to the University of Wisconsin by U.S. DOE Grant No. DE-FG52-06NA26196.

1:13PM R13.00002 Simultaneous Measurements of Density and Velocity Fields in Single-interface Richtmyer-Meshkov Instabilities , RICARDO MEJIA-ALVAREZ, BRANDON WILSON, KATHY PRESTRIDGE, Los Alamos National Laboratory, EXTREME FLUIDS TEAM — The Extreme Fluids Team at Los Alamos National Laboratory (LANL) has developed a new Vertical Shock Tube (VST). This facility is equipped with high-resolution diagnostics for simultaneous measurements of density and velocity fields in single-interface Richtmyer-Meshkov instabilities (RMI). The VST was conceived to provide high resolution PIV and PLIF data for a better understanding of the underlying phenomena of single-interface RMI. Additionally, these results will serve as a benchmark for RANS models and ILES. To reduce downtime and improve repeatability, a membraneless driver operates this VST. However, the Mach number response to the driver pressure differs from the one in membrane-based shock-tubes. Such differences are addressed in this talk. Additionally, this VST has the capability of introducing multi-modal 3-D perturbations in the interface between the working gases. Some examples of a perturbed air/SF_6 interface are presented. Finally, an instance of a simultaneous PIV-PLIF measurement at the initial conditions is presented.

1:26PM R13.00003 Investigating Mach number dependence on Richtmyer-Meshkov mixing with high resolution velocity and density measurements , GREG ORLICZ, SRIDHAR BALASUBRAMANIAN, KATHY PRESTRIDGE, Los Alamos National Laboratory, EXTREME FLUIDS TEAM — Experiments are performed to study the effect of incident shock Mach number (M) on the development of the Richtmyer-Meshkov instability after a shock wave impulsively accelerates a varicose-perturbed, heavy-gas curtain (air-SF_6-air). Incident shock strength is varied within the weak shock regime ($M \leq 1.5$), and the resulting instability and subsequent fluid mixing is measured using simultaneous Planar Laser-Induced Fluorescence (PLIF) and Particle Image Velocimetry (PIV). While large scale features of the evolving layer (e.g. total width) tend to grow similarly in scaled time, differences are observed in how the mixing occurs at smaller length scales. Presented are measures of mixing that can quantify some of these differences, as well as provide calibration and validation data for numerical models of this transitional flow. Through measurements of density PDFs, the instantaneous mixing rate, the density self-correlation parameter, and the area of the mixing layer, it is concluded that for fixed initial conditions, as incident shock Mach number is increased, the uniformity of the mixing layer increases, but the total amount of mixing decreases.

1:39PM R13.00004 Turbulent Mixing in Late-Time Richtmyer-Meshkov Instability Experiments , CHRIS WEBER, NICK HAEHN, JASON OAKLEY, DAVID ROTHAMER, RICCARDO BONAZZA, University of Wisconsin — The Richtmyer-Meshkov instability is experimentally investigated in a vertical shock tube using a broadband initial condition imposed on an interface between a helium-acetone mixture and argon (A= 0.7). The initial condition is created, first by setting up a gravitationally stable stagnation plane between the gases, and then injecting the same two gases horizontally at the interface to create a shear layer. The perturbations along the shear layer create a statistically-repeatable broadband initial condition. The interface is accelerated by a $M = 1.6$ or $M = 2.2$ planar shock wave and develops into a fully-developed turbulent state. Mixing measurements are made using planar laser-induced fluorescence (PLIF). The spectra, length scales, and isotropy after the turbulent mixing transition are presented.

1:52PM R13.00005 Evolution of the density self-correlation in developing RM turbulence , CHRISTOPHER TOMKINS, B.J. BALAKUMAR, G. ORLICZ, K. PRESTRIDGE, J.R. RISTORCELLI, Los Alamos National Laboratory — Turbulent mixing in a Richtmyer-Meshkov unstable light-heavy-light (air-SF_6-air) fluid layer subjected to a shock (Mach 1.20) and a reshock (Mach 1.17) is investigated using true ensemble statistics obtained from simultaneous velocity-density measurements. The mixing is found to be driven by an unstable array of initially symmetric vortices that induce rapid material mixing and create smaller scale vortices. The density self-correlation (b) and terms in its evolution equation are directly measured experimentally for the first time after reshock. Amongst other things, it is found that production terms are balanced by the dissipation terms, suggesting a form of equilibrium in b. Simultaneous velocity measurements are used to probe the state of the incipient turbulence. Results suggest that an inertial range is just beginning to form, consistent with the onset of a mixing transition. Second-order structure functions of the density field do not exhibit the classical 2/3 power-law behavior, which is discussed.

2:05PM R13.00006 Evaluation of the Predictive Capability of a Reynolds-Averaged Navier-Stokes Model Applied to Reshocked Richtmyer-Meshkov Instability[1] , TIBERIUS MORAN-LOPEZ, JAMES P. HOLLOWAY, University of Michigan, OLEG SCHILLING, Lawrence Livermore National Laboratory — Reshocked Richtmyer–Meshkov turbulent mixing of sulfur hexafluoride and air for 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. Mixing layer widths from simulations with Mach number $Ma = 1.45$ and Atwood number $At = -0.67$ are compared to the experimental data of Poggi, Thorembey and Rodriguez, and widths from simulations with $Ma = 1.24$, 1.50, and 1.98 with $At = 0.67$ are compared to the experimental data of Vetter and Sturtevant. The sensitivity of the mixing layer widths to variations in the initial conditions and key model coefficients is considered. Budgets of the turbulent transport equations are also considered to further elucidate the mechanisms contributing to turbulent mixing in reshocked Richtmyer–Meshkov instability experiments.

[1]This work was funded by the U. S. DOE NNSA under the Predictive Science Academic Alliances Program by grant DE-FC52-08NA28616 and performed under the auspices of the DOE by Lawrence Livermore National Laboratory under Contract DE-AC52-07NA27344.

2:18PM R13.00007 Simulations of Material Mixing in a Laser-Driven Reshock Experiment , BRIAN HAINES, FERNANDO GRINSTEIN, LESLIE WELSER-SHERRILL, JAMES FINCKE, Los Alamos National Laboratory — We perform simulations of a laser-driven reshock experiment in order to better understand material mixing driven by the Richtmeyer-Meshkov instability. Due to high sensitivities to target imperfections in the experimental data, direct comparisons of simulation and x-ray data are insufficient for validation. Therefore, we supplement these comparisons by performing spectral analysis. We also compare statistics of the data to results from DNS and theory of homogeneous isotropic turbulence. Our results show that in shock-driven transitional flows, some turbulent features, such as self-similarity and isotropy, only fully develop once others have decayed significantly. Finally. we evaluate a presumed PDF model for mixing at subgrid scales.

2:31PM R13.00008 Comparative Study of the Predictions of Four-Equation Reynolds-Averaged Navier-Stokes Models Applied to Richtmyer-Meshkov Instability-Induced Mixing[1] , OLEG SCHILLING, Lawrence Livermore National Laboratory — A multicomponent, weighted essentially nonoscillatory implementation of several four-equation K-ϵ and K-L based Reynolds-averaged Navier–Stokes models is used to simulate reshocked Richtmyer–Meshkov turbulent mixing at various Mach and Atwood numbers. One class of models is based on mechanical turbulence coupled to scalar variance and its dissipation rate, and the other is based on mechanical turbulence coupled to mass flux and the density–specific volume correlation. The predicted evolution of the mixing layer, molecular mixing and other quantities obtained from these models are systematically intercompared, as well as compared to experimental shock tube data. The relative advantages and disadvantages of the various models are discussed.

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

2:44PM R13.00009 Numerical investigation of three-dimensional effects in Richtmyer-Meshkov induced mixing processes , NIKOLAUS ADAMS, VOLKER TRITSCHLER, Technische Universität München, Institute of Aerodynamics and Fluid Mechanics — The Richtmyer-Meshkov instability (RMI) occurs when a perturbed interface between two fluids with different densities is impulsively accelerated by a passing shock wave. The misalignment of pressure gradient and density gradient between the fluids upon shock passing causes baroclinic vorticity production. The deposited vorticity drives the primary instability that causes the small initial perturbations at the interface to grow. In two-dimensional simulations of RMI the vortex stretching and tilting term vanishes and vorticity is confined to be perpendicular to the flow. However, for realistic Richtmyer-Meshkov induced mixing processes the vortex stretching term can be essential. The assessment of three-dimensional effects on RMI is the objective of our numerical investigation. We report on simulations of three-dimensional Navier-Stokes simulations of shock-cylinder interaction with re-shock. A SF6-gas cylinder is impacted by a Mach 1.2 shock wave propagating in air. The cylinder surface has an initial sinusoidal single-mode perturbation in the axial direction. The initial surface perturbation triggers an instability and vorticity evolution in all three space dimensions. Three-dimensional effects are further reinforced by the re-shock.

2:57PM R13.00010 High Resolution Numerical Investigation of Turbulence in a Reshocked Richtmyer Meshkov Unstable Curtain of Dense Gas[1] , SANTHOSH SHANKAR, SANJIVA LELE, Stanford University — High resolution numerical simulation of the impulsive acceleration of a dense gas curtain in air by a Mach 1.21 planar shock (modeling the experiments by Balakumar et al. PoF 2008) is carried out by solving the 3-D compressible multi-species Navier-Stokes equation coupled with a localized artificial diffusivity method to capture discontinuities in the flow-field. The simulations account for the presence of three species in the flow-field: air, SF_6 and acetone (used as a tracer species in the experiments). The reshock process is studied by re-impacting the evolving curtain with a reflected shock wave. Turbulence statistics computed in the flow-field following reshock are reported and compared with experiment where possible. Inertial range scaling, vorticity anisotropy and Reynolds stress development are studied in the reshocked flow. The high resolution data set is used to test certain modeling assumptions appearing in mixing models (BHR model) that have been traditionally used to study variable density flows.

[1]The work is supported by DOE-SciDAC grant DE-FC02-06-ER25787. Computer time was provided by ALCF through the INCITE award.

Tuesday, November 20, 2012 1:00PM - 3:10PM –

Session R14 Rotating Flows III 27B - Chair: Susan Kurien, Los Alamos National Laboratory

1:00PM R14.00001 Flow Characterization in a Spinning Film Apparatus[1] , ALONSO ALVARADO-SAVARAIN, ELLEN LONGMIRE[2], University of Minnesota- Aerospace Engineering & Mechanics — Flow generated in a mixing apparatus with similarities to but distinct deviations from a standard Taylor-Couette geometry is examined. Here an inner cylinder rotates about a vertical axis as an impeller within a stationary outer cylinder. The radius ratio is 0.95 and the aspect ratio of outer cylinder length to gap width is 27.5. The inner cylinder is hollow and shorter than the outer cylinder, leaving a bottom gap of 2.5 times the inter-cylinder gap width. The apparatus volume is partially full of liquid such that an inner free surface forms during operation. Velocity statistics in the side and bottom gaps are determined by laser Doppler velocimetry for characteristic Taylor (or Reynolds) numbers based on gap width in the range 1100-4700 which represent wavy vortex and turbulent regimes in Taylor-Couette flow. Experimental results for the aforementioned Reynolds numbers in conditions where the liquid present in the system is 30%, 43% and 52% of the total volume are shown. Additional results having the system modified to allow axial throughflow maintaining dimensions and liquid holdup equal to the batch conditions are also shown.

[1]Nano Dispersions Technology, Inc. and SENACYT-Panama (Scholarship for professional excellence)
[2]Fellow, American Physical Society, 2007; NSF National Young Investigator, 1994 - 1999

1:13PM R14.00002 Layer formation in rotating and stratified flows , SUSAN KURIEN, Los Alamos National Laboratory, LESLIE SMITH, University of Wisconsin, Madison — We present a numerical study of layer formation in forced, rotating, stably stratified Boussinesq flows. We focus on parameter regimes with buoyancy frequency N and rotation frequency f chosen such that the timescales $1/N$ and $1/f$ are at least as fast as the nonlinear timescales. The aspect-ratio of the domain is $\delta = H_d/L_d$ where H_d and L_d are the domain height and width respectively. Two sets of calculations are studied at small, nearly fixed Froude number $Fr = U/(HN) \approx 0.002$ where H is fixed at one-quarter of H_d and U is the characteristic forcing based velocity scale. The first set fixes $\delta = 1$ with N/f values ranging from 1 to 32. The second set fixes the Burger number $Bu = \delta N/f = 1$ with aspect ratio $\delta = H_d/L_d$ ranging from 1 to 1/16. We show that both rotation rate and domain aspect-ratio conspire to set the scale and structure of the layers formed in the flows.

1:26PM R14.00003 Density Stratification of Rotating Flow in Coaxial Cylinder , SUNGSU LEE, HYUN AH SON, Chungbuk National University, ALBERT S. KIM, University of Hawaii at Manoa — Flows containing particulates often pose problems in many engineering practices among which the separation of clean water from polluted solution is essential in environmental engineering. In this study, the stratification of density in a coaxial cylinder are investigated using computational fluid dynamics with Boussinesq approximation. Particulate flow is injected into the domain between the vertical coaxial cylinders and allowed to leave the domain through the exit located upper part of the inner cylinder. During the rotation initiated by the tangential momentum, the hydrodynamic forces stratifies the particulate flow and interact with gravitational force as well as friction, which make the flow instable and complex. This study includes parametric investigation by varying the density of the particulates in the flow and the size of the inner cylinder. The results will present the effectiveness of the stratification which corresponds to the vortex separation in many engineering practice. This work was financially supported by projects of the "Development of Energy utilization technology with Deep Ocean Water," KIOST of Korea.

1:39PM R14.00004 Transitions in turbulent plane Couette flow with rotation[1] , MATTHEW SALEWSKI[2], BRUNO ECKHARDT, Fachberiech Physik, Philipps-Universitaet Marburg — The interplay of shearing and rotational forces in fluids significantly affects the transport properties of turbulent fluids such as the heat flux in rotating convection and the angular momentum flux in a fluid annulus between differentially rotating cylinders. A numerical investigation was undertaken to study the role of these forces using plane Couette flow subject to rotation about an axis perpendicular to both wall-normal and streamwise directions. Using a set of progressively increasing Reynolds numbers ($650 \leq Re \leq 5200$), our primary findings show the momentum transport for a given Re is a smooth but non-monotonic function of inverse Rossby number ($1/Ro$). For lower turbulent Reynolds numbers, $Re \leq 1300$, a peak in momentum transport occurs at $1/Ro = 0.2$; this peak is 50% higher than the non-rotating ($1/Ro = 0$) flux and is attributed to the turbulent Taylor vortices. However, as the shear is increased to $Re = 5200$, a second stronger peak emerges at $1/Ro = 0.03$. The flux at the second peak is nearly 20% larger than the non-rotating flux compared to the Taylor vortex peak which is now only 16% larger. This finding contributes to the understanding of the torque maximum found in the high-turbulence Taylor-Couette experiments in Maryland, USA and Twente, NL.

[1]Funded by the German Research Foundation (DFG), Project FOR-1182
[2]Alternative email: matthewsalewski@gmail.com

1:52PM R14.00005 Stability and transition in rotating plane Couette flow[1] , CONOR DALY, NIGEL PEAKE, University of Cambridge — It is well known that the addition of spanwise Coriolis rotation to plane Couette shear flow causes the laminar solution to destabilise. Linear stability theory predicts a secondary solution of steady, streamwise oriented vortices, with numerical and experimental evidence in support of the predicition. We compute the linear stability of the secondary solutions, and find that the characteristics of the least stable mode depend on the non-dimensionalised rotation parameter. We discuss the transition phenomena arising from the different secondary instabilities, which lead to a range of steady and unsteady tertiary states.

[1]This work is funded by EPSRC.

2:05PM R14.00006 Rotating plane Couette flow at high rotation number , A. SURYADI, N. TILLMARK, P.H. ALFREDSSON, Linne FLOW Centre, Stockholm, SWEDEN — Flow structures in the rotating plane Couette flow facility at KTH (described in Tsukahara, et al. *J. Fluid Mech.* vol. **648**) have been studied at high rotation numbers. The test section is 20 mm wide with a length of 1500 mm in the streamwise (x) and 360 mm in the spanwise (z) directions and can be rotated in the spanwise direction up to angular velocities of $\Omega_z \approx 0.6$ rad/s. The flow is characterised by: (1) the Reynolds number Re based on the test section's half-width (h) and half of the velocity difference between the moving walls, (2) the rotation number $\Omega = 2\Omega_z h^2/\nu$. For low rotation numbers the primary instability consists of streamwise-oriented roll cells, but Tsukahara, et al. showed the secondary instability in the form of wavy streamwise oriented roll-cells at $Re = 100$ and $\Omega = 3 - 12$, whereas for higher Ω, the flow structures again stabilize to streamwise-oriented roll cells. Here we find that at even higher Ω in the range $40 - 70$, a new type of secondary instability develops in the form of counter-rotating helical roll-cells. The structure of this instability, as well as other instabilities, are investigated by flow visualization as well as two-dimensional PIV-measurements in several xz-planes.

2:18PM R14.00007 Nonlinear evolution of the elliptical instability , ADRIAN BARKER, YORAM LITHWICK, Northwestern University — Tidal interactions between short-period gaseous planets and their host stars can have important effects on the orbits of the planets. In particular, the observational preponderance of circular orbits amongst short-period planets, relative to those in wider orbits, is thought to be explained by tidal dissipation in the planets. However, the mechanisms responsible for this are poorly understood. To a first approximation, the linear response of a rotating gaseous planet to the tidal gravitational perturbation of its host star is an elliptical flow, with its major axis aligned with the star. This flow is subject to the elliptical instability, which is a generic linear instability of elliptical streamlines. We will discuss results from a set of high-resolution numerical simulations of the nonlinear evolution of the elliptical instability in a local model of a rotating tidally deformed fluid body. This model allows a detailed study of the dissipative properties of the turbulence driven by the elliptical instability. We will also present results of simulations including the effects of a weak magnetic field, and how this modifies the resulting evolution. The importance of this mechanism for explaining the observations will be discussed.

2:31PM R14.00008 Experimental evidence of a triadic resonance of plane inertial waves in a rotating fluid , THIERRY DAUXOIS, GUILHEM BORDES, Laboratoire de Physique de l Ecole Normale Superieure de Lyon, CNRS and Universite de Lyon, France, FRÉDÉRIC MOISY, PIERRE-PHILIPPE CORTET, Laboratoire FAST, CNRS, Univ Paris Sud, UPMC Univ Paris 06, France, STRATIFIED FLUIDS TEAM, ROTATING FLUIDS TEAM — Plane inertial waves are generated using a wavemaker, made of oscillating stacked plates, in a rotating water tank. Using particle image velocimetry, we observe that, after a transient, the primary plane wave is subject to a subharmonic instability and excites two secondary plane waves. The measured frequencies and wavevectors of these secondary waves are in quantitative agreement with the predictions of the triadic resonance mechanism. The secondary wavevectors are found systematically more normal to the rotation axis than the primary wavevector: this feature illustrates the basic mechanism at the origin of the energy transfers towards slow, quasi two-dimensional, motions in rotating turbulence.

Reference: G. Bordes, F. Moisy, T. Dauxois, P.-P. Cortet, Phys. Fluids **24**, 014105 (2012), doi: 10.1063/1.3675627

2:44PM R14.00009 Earth rotation prevents exact solid body rotation of fluids in the laboratory , PIERRE-PHILIPPE CORTET, JEAN BOISSON, Laboratoire FAST, CNRS, Univ Paris Sud, UPMC Univ Paris 06, France, DAVID CÉBRON, Institut fur Geophysik, ETH Zurich, Switzerland, FRÉDÉRIC MOISY, Laboratoire FAST, CNRS, Univ Paris Sud, UPMC Univ Paris 06, France, ROTATING FLUID TEAM, EARTH AND PLANETARY MAGNETISM TEAM — We report direct evidence of a secondary flow excited by the Earth rotation in a water-filled spherical container spinning at constant rotation rate. This so-called *tilt-over flow* essentially consists in a rotation around an axis which is slightly tilted with respect to the rotation axis of the sphere. In the astrophysical context, it corresponds to the flow in the liquid cores of planets forced by precession of the planet rotation axis, and it has been proposed to contribute to the generation of planetary magnetic fields. We detect this weak secondary flow using a particle image velocimetry system mounted in the rotating frame. This secondary flow consists in a weak rotation, thousand times smaller than the sphere rotation, around a horizontal axis which is stationary in the laboratory frame. Its amplitude and orientation are in quantitative agreement with the theory of the tilt-over flow excited by precession. These results show that setting a fluid in a perfect solid body rotation in a laboratory experiment is impossible — unless tilting the rotation axis of the experiment parallel to the Earth rotation axis.
Reference: J. Boisson, D. Cébron, F. Moisy and P.-P. Cortet, EPL **98**, 59002 (2012), doi: 10.1209/0295-5075/98/59002

2:57PM R14.00010 A fluid Foucault pendulum: the impossibility of achieving solid-body rotation on Earth , ROBERT BLUM, DANIEL ZIMMERMAN, SANTIAGO TRIANA, DANIEL LATHROP, University of Maryland, College Park — Rotating fluid dynamics is key to our understanding of the Earth's atmosphere, oceans, and core, along with a plethora of astrophysical objects. Laboratory study of these natural systems often involves spinning experimental devices, which are assumed to tend to rigid rotation when unstirred. We present results showing that even at the tabletop scale, there is a measurable oscillatory flow driven by the precession of the experiment's axis as the earth rotates. We measure this flow in a rotating cylinder with an adjustable aspect ratio. The horizontal flow in the rotating frame is measured using particle tracking. The steady state is well-described by an inertial mode whose amplitude is maximum when the height to diameter ratio is 0.995, which matches theoretical predictions. We also quantify the resonant amplitude of the inertial mode in the cylinder and estimate the amplitude in other devices. We compare our results to similar studies done in spherical devices. [Triana et al, JGR, 117 (2012), B04103][Boisson et al, EPL, 98 (2012), 59002]

Tuesday, November 20, 2012 1:00PM - 3:10PM –

Session R15 Biofluids: Flying and Swimming 28A - Chair: Philippe Bardet, The George Washington University

1:00PM R15.00001 Flow structures in the wake of heaving and pitching foils , DEREK NAJDZIN, ENRIQUE PARDO, MEGAN C. LEFTWICH, PHILIPPE M. BARDET, The George Washington Univeristy — A 10-bar mechanism drives a cambering hydrofoil in an oscillatory heaving and pitching motion that replicates the flapping motion of a dolphin tail. The mechanism sits on a force-balance with six strain gages that together measure the forces and moments experienced by the fin during an oscillation. Planar Laser-Induced Fluorescence is used to image the flow structures created downstream of the cambering fin for a range of Reynolds and Strouhal numbers. The images are taken in the mid-plane, parallel to the bottom of the water tunnel. These results are compared to a rigid foil at matching conditions to investigate the role of camber changes during the flapping cycle.

1:13PM R15.00002 Force measurement in heaving and pitching foils , ENRIQUE PARDO, DEREK NAJDZIN, MEGAN C. LEFTWICH, PHILIPPE M. BARDET, The George Washington University — This study analyzes the efficiency of a cambering hydrofoil built to simulate the movement of flukes on cetaceans. The mechanism is a 10 bar assembly that allows a hydrofoil to move in a cambered pitching and heaving motion similar to that of a dolphin. The mechanism sits on a force-balance with six strain gages that together measure the three forces and three moments experienced by the fin during a cycle of motion. These gages are attached to a traveling mechanism that rest on a water flume. To analyze the efficiency of the hydrofoil we took measurements at various Reynolds and Strouhal numbers. These measurements were done twice were compared to the thrust produced by a rigid (non-cambered) hydrofoil at the same conditions.

1:26PM R15.00003 3D Synthetic Aperture PIV of a Freely Swimming Fish , LEAH MENDELSON, ALEXANDRA TECHET, Massachusetts Institute of Technology — Fish owe much of their locomotive success to complex body geometries and wake interactions that cannot be fully characterized by planar experimental techniques including 2D PIV. Volumetric methods are valuable to illustrate and quantify these features, thus providing new insights for bioinspired design. In particular, synthetic aperture particle image velocimetry (SAPIV) uses light field imaging algorithms to reconstruct a 3D particle field which can then be analyzed using a 3D cross-correlation. Previous studies have shown that this technique is able to resolve all three velocity components on the same order length scale and to see around partial occlusions, such as a caudal fin, through the use of multiple cameras. To harness these capabilities for biomimetic use, SAPIV is used to depict the three-dimensional velocity field and vortical structures surrounding a freely swimming Giant danio (Devario aequipinnatus) during straight swims and turning maneuvers.

1:39PM R15.00004 Hydrodynamic performance of multiple bodies swimming in an in-line configuration[1] , BIRGITT BOSCHITSCH, PETER DEWEY, ALEXANDER SMITS, Princeton University — Experiments are reported on a pair of airfoils that are harmonically pitched about their leading edges and arranged in an in-line configuration to determine the hydrodynamic effect of drafting behind a neighbor in unsteady bio-inspired propulsion. The thrust production, power consumption, and propulsive efficiency is independently measured for the leading and trailing airfoils at a Reynolds number of 2000 for a range of streamwise airfoil spacings, Strouhal numbers, and oscillation phase differential between the airfoils. To assess the wake interactions between the panels that lead to propulsive performances observed, digital particle image velocimetry (DPIV) is used. These results are compared to an airfoil swimming in an isolated configuration to identify the parameters that lead to a benefit (or detriment) when swimming in-line with a neighbor.

[1]The authors would like to acknowledge the generous support from the Office of Naval Research under Program Director Dr. Bob Brizzolara, MURI grant number N00014-08-1-0642.

1:52PM R15.00005 Dynamics of a heaving flexible foil in a uniform flow , FLORINE PARAZ, CHRISTOPHE ELOY, LIONEL SCHOUVEILER, IRPHE — Most aerial and aquatic animals produce thrust using flapping flexible appendages. The performances of such propulsion systems are strongly related to the appendages dynamics, in particular to the amplitude of the trailing edge motion and to the vortical patterns produced. A better understanding of this mode of propulsion requires to investigate the dynamics of the flexible appendages, as a response to harmonic forcing. In this context, experiments are performed with flexible foils immersed in the uniform flow of a water channel. A harmonic heaving motion, that is transverse to the foil, is then imposed to its leading edge. The response of the foil likely results from the resonance between the forcing and the natural modes of vibration. Experimental results are compared with a two-dimensional model assuming a zero-thickness flexible sheet of infinite span immersed in a potential flow.

2:05PM R15.00006 Experimental investigation of 2D flexible plunging hydrofoil[1] , RUIJUN TIAN, ROBERT MITCHELL, FANGJUN SHU, New Mexico State University — It is believed that both birds and insects benefit from their wing flexibility during the flapping flight. One of the possible benefits is higher lift force generation capability than that of rigid wing models. Both experimental and computational work has discovered that the leading edge vortex (LEV) plays an important role in this advantage of high lift force generating efficiency. In the present work, flow physics related to high lift-generating flexible wings are investigated experimentally. Both flexible and rigid hydrofoils (NACA0012) were actively plunged in glycerol-water solution with various amplitude, frequency and Reynolds number combinations. Phase-locked Particle Image Velocimetry (PIV) measurements were conducted to investigate the generation and evolution of the LEVs. Lift and drag forces during plunging were also measured to uncover the relationship between the force response and the surrounding flow field development. The overall results were also compared between flexible and rigid hydrofoils to provide qualitative data for validation of computational work.

[1]Supported by Army High Performance Computing Center.

2:18PM R15.00007 Performance of a wing with nonuniform deformability in hovering flight[1] , KOUROSH SHOELE, RE Vision Consulting, LLC, QIANG ZHU, UC San Diego — The deformability of insect wings are related to the embedded skeleton (venation). In this study, the aerodynamic performance of wings with nonuniform flexibility is computationally investigated. By using a two-dimensional rendition, the underlying veins are modeled as springs, and the membrane is modeled as a flexible plate. The focus is on the effects of the detailed distribution of vein flexibility upon the performance of such a wing in the generation of lift force. Specifically, we are interested to find the importance of leading edge strengthening. Towards this end, the aerodynamic performances of three wings, a rigid wing, a flexible wing with identical veins, and a flexible wing with strengthened leading edge, are studied and compared against each other. It is shown that the flexible wing with leading edge strengthening is capable of producing significantly higher lift force. This is found to be related to the cambering effect at the leading edge, which enhances the leading edge vortices. In addition, in contrast to the other two wings, which show sensitivity to kinematic parameters, the wing with strengthened leading edge perform well over a wide range of parameters.

[1]This work is supported by NSF under grant CBET0844856.

2:31PM R15.00008 Rotational inertial effects on flexible wing , DEWEI QI, Western Michigan University, RAYMOND GORDNIER, AFRL/RBAC, Wright Patterson AFB OH 45435 — To understand rotational inertial effects on aerodynamic force, the lattice Boltzmann flexible particle method (LBFPM) is employed to simulate interaction between fluid flows and flapping motion of a chord-wise flexible wing in a 3D space at two levels of pitching or rotational rates corresponding to two rotational Reynolds numbers of Rer=356 and 107 while the translation Reynolds number is kept at the same level of Re=136. At each rotational Reynolds number, flexibility and mass ratio of wing to fluid are systematically varied at different levels and lift, drag, deformation and power efficiency are computed and compared. It is found that the lift force and power efficiency increase non-linearly up to maximum values as chord-wise flexibility increases, then fall down as flexibility continuously increases for the larger rotational Reynolds number of Rer=356. As the mass ratio increases the inertial force and the lift force increase while the input power increases. The flexibility should be optimized by the lift force and the power efficiency. The simulation results indicate that rotational inertia is an important factor for flexibility to enhance lift and power efficiency. However, the case with a lower rotational Reynolds number of Rer=107 does not have this behavior. It is also found that the deflection angle and the sweeping distance in the vertical direction are much larger for trailing edge than leading edge.

2:44PM R15.00009 Sectional lift coefficient of a rotating wing at low Reynolds number[1] , JIEUN KIM, JIHOON KWEON, HAECHEON CHOI, Seoul National University — We investigate the characteristics of sectional lift force on a rotating wing at low Reynolds number using three-dimensional numerical simulation. Three different types of flat plate wings (fruit-fly, rectangular and triangular wings) are considered but keeping their aspect ratio (wing span/wing chord) the same at 3.74. The wings rotate at a constant angular velocity and the angle of attack is fixed during rotation ($5^\circ \sim 45^\circ$). The Reynolds number is 136 based on the wing chord length and the translational velocity at the wing tip, corresponding to that of the flapping fruit-fly wing in hovering flight. An immersed boundary method in a non-inertial reference frame (Kim and Choi, JCP, 2006) is used to simulate the flow. During the first rotation, the sectional lift coefficient decreases from the wing root to the wing tip for all cases. After several rotations, however, the sectional lift coefficient becomes nearly constant except near the wing root and tip at low angles of attack ($\leq 15^\circ$), but maintains a similar behavior to that of first rotation at high angle of attack ($\sim 45^\circ$). Finally, the wing shape does not significantly change the spanwise distribution of sectional lift coefficient.

[1]Supported by the NRF Program (2011-0028032).

2:57PM R15.00010 Aerodynamic effect of alula in avian flight[1] , SANG-IM LEE, JAEMYOUNG LEE, HYUNGMIN PARK, PIOTR JABLONSKI, HAECHEON CHOI, Seoul National University — Alula is a small structure located at the joint between handwing and armwing of birds and has been suggested to function as a leading-edge slot. In this study, we investigated the functional aspect of alula in bird flight with experimental conditions that reflect the flow characteristics used by birds in their actual flight using magpies as the model species. The presence of alula enabled the bird to perform steeper descending flights with greater lateral angle changes. Force measurements showed that alula presence increased the lift when the angle of attack was high (higher than 20-45 deg), which resulted in the stall delay by 5 deg. The wake width was significantly thinner when alula was present, suggesting that boundary layer separation is delayed when alula is used. This result was corroborated by PIV; accelerated streamwise velocity over the wing surface was recovered faster and separation point was pushed downstream when alula was present. To conclude, the lift enhancement and stall delay by alula are closely related to the downstream movement of separation point and faster recovery of accelerated flow over the wing surface, which endows greater flight maneuverability to the birds.

[1]This work was supported by the Korea Research Foundation Grants (2011-0030744, 2010-0009006, and 2012-K001368).

Tuesday, November 20, 2012 1:00PM - 2:44PM –

Session R16 Biofluids: Biofilms and Membranes 28B - Chair: Howard Stone, Princeton University

1:00PM R16.00001 Biofilm streamers cause rapid clogging of flow systems , YI SHEN, KNUT DRESCHER, NED WINGREEN, BONNIE BASSLER, HOWARD STONE, Princeton Universtiy — Biofilms are antibiotic-resistant, sessile bacterial communities that are found on most surfaces on Earth. In addition to constituting the most abundant form of bacterial life, biofilms also cause chronic and medical device-associated infections. Despite their importance, basic information about how biofilms behave in common ecological environments is lacking. Here we demonstrate that flow through soil-like porous materials, industrial filters, and medical stents dramatically modifies the morphology of *Pseudomonas aeruginosa* biofilms to form streamers which over time bridge the space between obstacles and corners in non-uniform environments. Using a microfluidic model system we find that, contrary to the accepted paradigm, the accumulation of surface-attached bacterial biofilm has little effect on flow resistance whereas the formation of biofilm streamers causes sudden and rapid clogging. The time at which clogging happens depends on bacterial growth, while the duration of the clogging transition is driven by flow-mediated transport of bacteria to the clogging site. Flow-induced shedding of extracellular matrix from the resident biofilm generates a sieve-like network that catches bacteria flowing by, which add to the network of extracellular matrix, to cause exponentially rapid clogging. We expect these biofilm streamers to be ubiquitous in nature, and to have profound effects on flow through porous materials in environmental, industrial, and medical environments.

1:13PM R16.00002 The "Swiss cheese" instability of bacterial biofilms , HONGCHUL JANG, ROBERTO RUSCONI, ROMAN STOCKER, MIT — Bacteria often adhere to surfaces, where they develop polymer-encased communities (biofilms) that display dramatic resistance to antibiotic treatment. A better understanding of cell detachment from biofilms may lead to novel strategies for biofilm disruption. Here we describe a new detachment mode, whereby a biofilm develops a nearly regular array of ∼50-100 μm holes. Using surface-treated microfluidic devices, we create biofilms of controlled shape and size. After the passage of an air plug, the break-up of the residual thin liquid film scrapes and rearranges bacteria on the surface, such that a "Swiss cheese" pattern is left in the residual biofilm. Fluorescent staining of the polymeric matrix (EPS) reveals that resistance to cell dislodgement correlates with local biofilm age, early settlers having had more time to hunker down. Because few survivors suffice to regrow a biofilm, these results point at the importance of considering microscale heterogeneity in assessing the effectiveness of biofilm removal strategies.

1:26PM R16.00003 (How) do biofilms control their morphology? , AGNESE SEMINARA, NAVEEN SINHA, JAMES WILKING, DAVID WEITZ, MICHAEL BRENNER, Harvard SEAS — Bacterial biofilms are organized communities of cells living in association with surfaces. The hallmark of biofilm formation is a well defined spatio-temporal pattern of gene expression, leading to differentiation and a complex morphology. While this process resembles the development of a multicellular organism, biofilms are only transiently multicellular. More importantly the functions associated to the biofilm phenotype are largely unknown. Here we discuss aspects of biofilm physiology connected to motility and nutrient uptake. We develop a connection between patterns of gene expression and morphology and finally we propose a framework to understand how these gene expression patterns may be generated and possibly controlled.

1:39PM R16.00004 Morphological Approach toward Elucidating Transport and Shear Behavior of Biofilms , ALOKE KUMAR, Biosciences Division, Oak Ridge National Lab, Oak Ridge, TN 37831, PALLAB BARAI, PARTHA MUKHERJEE, Department of Mechanical Engineering, Texas A&M University, College Station, TX 77843 — Biofilms are complex three-dimensional matrix encapsulated aggregations of microbes that grow on a solid surface. Distribution of microbes inside the matrix or extracellular polymer substances (EPS) affects diffusive transport as well as mechanical response of the biofilm under shear induced deformation. In this work, a morphology-aware computational approach encompassing a digital representation of the biofilm is presented. Confocal microscopy images of biofilms are employed for the digital morphology constructs. For mechanical response under shear, the biofilm can be viewed as rigid bacteria inclusions dispersed inside a cross-linked polymer gel (EPS). The digital biofilm model takes into account the unfolding behavior of proteins to characterize the mechanical response of the EPS. Experimentally observed strain stiffening behavior of biofilms has been captured using the computational approach. Transport simulation reveals the influence of bacterial loading and aggregates in the biofilm on the diffusion behavior.

1:52PM R16.00005 Reflection and refraction of flexural waves in membranes with complex geometry , ARTHUR EVANS, University of California, Los Angeles, BASANTA BHADURI, RYAN TAPPING, GABRIEL POPESCU, University of Illinois, Urbana-Champaign, ALEX LEVINE, University of California, Los Angeles — Undulatory waves on membranes are studied in a variety of contexts including microrheology of red blood cell membranes, giant vesicles, and various cellular mimics, such as actin coated vesicles. While the fundamental understanding of undulatory dynamics in flat membranes is well known, the problem is significantly more interesting for waves on curved membranes, where geometry couples bending and stretching in the surface. In this talk we report on analysis of flexural wave dynamics in curved membranes and draw a useful analogy between the propagation of these waves and physical optics. We obtain an analog of Snell's law for the reflection and refraction of undulatory waves at interfaces at which the local mean and Gaussian curvature of the surface changes abruptly. In addition, we show that, due to the higher order derivatives in the force balance equation, bending waves on curved membranes generically exhibit characteristics associated with waves in classical optics, such as birefringence and total internal reflection. Using this latter insight, we analyze the experimentally observed spatial distribution of the amplitude of red blood cell membrane undulations, and show that one can understand their spatial structure in terms of the local geometry of the cell.

2:05PM R16.00006 Deformation and stability of biomimetic membranes in DC electric pulses , PAUL SALIPANTE, PETIA VLAHOVSKA, Brown University — Electrohydrodynamics of vesicles (closed bilayer membranes) made of lipids or polymers are investigated both experimentally and theoretically. 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. The evolution of vesicle shape, and in particular the oblate-prolate transition, is experimentally studied for DC pulses of different strength and durations. Membrane composition is varied to observe the effect of membrane viscosity, bending rigidity, and membrane capacitance. The results show that the transient response of the vesicle is sensitive to membrane viscosity, while the steady state shape is mainly controlled by membrane tension. Strong DC pulses, typically used in cell electroporation, induce an instability in both lipid and polymer membranes. The instability leads to vesicle collapse, where the timescale of collapse shows a $t \sim 1/E^2$ dependence.

2:18PM R16.00007 Influence of membrane viscosity on dynamics of capsules and red blood cells[1] , ALIREZA YAZDANI, PROSENJIT BAGCHI, Rutgers, The State University of New Jersey — Most previous continuum-level numerical studies on capsule and erythrocyte dynamics have ignored the role of membrane viscosity. We present a numerical method using a Kelvin–Voigt viscoelastic model for the capsule membrane. We observe that the membrane viscosity leads to buckling in the range of shear rate in which no buckling is observed for capsules with purely elastic membrane. For moderate to large shear rates, the wrinkles on the capsule surface appear in the same range of the membrane viscosity that was reported earlier for artificial capsules and red blood cells based on experimental measurements. It is also observed that the bending stiffness required to obtain stable shapes is also in the same range as that reported for the red blood cells, but considerably higher than that estimated for artificial capsules. Membrane viscosity is observed to reduce cell deformation, and introduce a damped oscillation in time-dependent deformation and inclination. The time-averaged inclination angle and the tank-treading frequency show nonmonotonic trends with increasing membrane viscosity. Further, the dynamics of a non-spherical capsule is observed to change from a swinging motion to a tumbling motion with increasing membrane viscosity.

[1]Funded by NSF

2:31PM R16.00008 Pair-collision between heterogeneous capsules in simple shear: Effect of membrane stiffness and membrane constitutive laws[1] , RAJESH SINGH, University of Delaware, KAUSIK SARKAR, George Washington University — Deformability of red blood cells affects hydrodynamic properties of blood and thereby physiological functions in many cardiovascular diseases, e.g. in sickle cell anemia and malaria, the cell membrane becomes stiff affecting their circulation through microvessels. Here, we numerically simulate the hydrodynamic interaction between a pair of cell-like capsules in a free shear flow, using a front-tracking method. The membrane is modeled using various constitutive equations. By varying the stiffness of one capsule (C_2) and keeping all other parameters constant, we find a significant effect on the deformation and trajectory of the other (C_1). Increasing the stiffness of C_2 surprisingly increases the peak deformation of C_1 while decreasing the cross-stream shift in its trajectory However, the relative trajectory between capsules remains the same. Effects of constitutive laws and difference in behaviors between capsules and drops are investigated explaining underlying physics.

[1]partial support from NSF

Tuesday, November 20, 2012 1:00PM - 3:10PM –

Session R17 Biofluids: Micro-swimming Theory II 28C - Chair: Jeffrey Guasto, Massachusetts Institute of Technology

1:00PM R17.00001 Studies of Ciliated Microorganism Swimming with and against a Magnetic Field Tuned Apparent Weight Force , ILYONG JUNG, HARRY MICKALIDE, JAMES M. VALLES JR., Brown University — There is a class of microorganisms that are small enough to swim at low Reynolds number but large enough for gravity to influence their behavior. Remarkably, *Paramecia* exert a stronger (weaker) propulsion force when swimming against (with) their apparent weight force, $\vec{W}$. To investigate the source of the swimming speed response, we are examining how the trajectories of single swimmers change when they reverse their direction relative to $\vec{W}$. We characterize their helical trajectories with three parameters that we can relate to their beating of their cilia using a simple model. The latest results will be described.

1:13PM R17.00002 Micropropulsion and microrheology in complex fluids via symmetry breaking , ON SHUN PAK, Department of Mechanical and Aerospace Engineering, University of California San Diego, LAILAI ZHU, LUCA BRANDT, KTH Mechanics, ERIC LAUGA, Department of Mechanical and Aerospace Engineering, University of California San Diego — Many biological fluids have polymeric microstructures and display non-Newtonian rheology. We take advantage of such nonlinear fluid behavior and combine it with geometrical symmetry-breaking to design a novel small-scale propeller able to move only in complex fluids. Its propulsion characteristics are explored numerically in an Oldroyd-B fluid for finite Deborah numbers while the small Deborah number limit is investigated analytically using a second-order fluid model. We then derive expressions relating the propulsion speed to the rheological properties of the complex fluid, allowing thus to infer the normal stress coefficients in the fluid from the locomotion of the propeller. Our simple mechanism can therefore be used either as a non-Newtonian micro-propeller or as a micro-rheometer.

1:26PM R17.00003 A Numerical Study of Muco-Ciliary Transport under the condition of Primary Ciliary Dyskinesia , PAHALA GEDARA JAYATHILAKE, Singapore-MIT Alliance, National University of Singapore, WAN LUNG LEE, None, DUC VINH LE, Institute of High Performance Computing, Singapore, HEOW PUEH LEE, BOO CHEONG KHOO, National University of Singapore — Primary ciliary dyskinesia (PCD) is a disease due to the defects in motile cilia. A two-dimensional numerical model based on the immersed boundary method coupled with the projection method is used for a preliminary study of the flow physics of muco-ciliary transport of human respiratory tract under PCD conditions. The effects of the cilia beating amplitude, cilia beat pattern (CBP), cilia beat frequency (CBF), immotile cilia, and uncoordinated beating of cilia on mucus transport are investigated. As expected, the mucus velocity decreases as the beating amplitude and CBF decrease. The windscreen wiper motion and rigid rod motion, which are two abnormal CBPs owing to PCD, would greatly reduce the mucus transport. The mucus velocity decreases rather linearly if the number of uniformly distributed immotile cilia increases. The results further show that the mucus velocity would be slightly reduced when the uniformly distributed immotile cilia are rearranged as a cluster of immotile cilia. Furthermore, if the half of the cilia are immotile and uniformly distributed, the incoordination between motile cilia would not significantly affect the mucus velocity.

1:39PM R17.00004 Dynamics of artificial bacterial flagella , YI MAN, ERIC LAUGA, Department of Mechanical and Aerospace Engineering, University of California, San Diego — Artificial bacterial flagella (ABF) are small-scale rigid helices actuated by an external rotating magnetic field and therefore able to propel in a viscous fluid. In experiments, ABF are observed to display wobbling motion at low frequencies and a transition to directed swimming at higher frequencies. We use here a combination of numerics and asymptotics to provide a theoretical explanation for this dynamics. In particular we show that the wobbling angle - the angle between the direction of propulsion and the axis of the helix - is inversely proportional to the Mason number, a dimensionless number given by the ratio of the magnitudes of viscous torque to magnetic torque. Our theoretical predictions agree well with experimental results.

1:52PM R17.00005 Analysis of the orbits of particles generating one-dimensional dynamic coherent structures , DENIS MELNIKOV, University of Brussels, DMITRI PUSHKIN, University of Oxford, VALENTINA SHEVTSOVA, University of Brussels, JEREMI COLLABORATION — The particle accumulation in time-periodic incompressible flows is studied experimentally and numerically. The geometry of interest is a cylindrical column of finite length where a liquid is suspended between two differentially heated horizontal flat disks. The increase of the temperature difference between disks beyond the critical values results in appearance of time periodic flows which are either standing or azimuthally travelling hydrothermal wave. In the case of travelling wave inertial particles have a tendency to spontaneously align in one-dimensional dynamic coherent structures called below as orbits. Three types of orbits are experimentally found in the system with polydisperse particles (differing in size, shape and density). Initial positions of different particles were close to each other. These orbits can be classified as periodic (loop PAS), quasi-periodic (toroidal PAS) and temporally synchronized orbits. Assuming the motion of particles is described by the simplified Maxey-Riley equation, the phenomenon of formation of particles accumulation structures was numerically analysed. The experimental observations are in a favourable agreement with the results of the computer simulations.

2:05PM R17.00006 Optimal shapes of surface-slip driven self-propelled swimmers , ANDREJ VILFAN, NATAN OSTERMAN, J. Stefan Institute — If one defines the swimming efficiency of a microorganism as the power needed to move it against viscous drag, divided by the total dissipated power, one usually finds values no better than 1%. In order to find out how close this is to the theoretically achievable optimum, we first introduced a new efficiency measure at the level of a single cilium or an infinite ciliated surface and numerically determined the optimal beating patterns according to this criterion [1]. In the following we also determined the optimal shape of a swimmer such that the total power is minimal while maintaining the volume and the swimming speed. The resulting shape depends strongly on the allowed maximum curvature. When sufficient curvature is allowed the optimal swimmer exhibits two protrusions along the symmetry axis. The results show that prolate swimmers such as Paramecium have an efficiency that is $\sim$ 20% higher than that of a spherical body, whereas some microorganisms have shapes that allow even higher efficiency.

[1] N. Osterman and A. Vilfan, Finding the ciliary beating pattern with optimal efficiency, Proc. Natl. Acad. Sci. USA, 108 15727-15732 (2011)

2:18PM R17.00007 Study of propulsion of microorganisms using viscous slender-body theory , SRIKANTH TOPPALADODDI, Department of Geology and Geophysics, Yale University, CT - 06511, USA, NEIL BALMFORTH, Department of Mathematics, University of British Columbia, Vancouver, BC, V6T 1Z2, Canada — In this study, we investigate the swimming of a slender axisymmetrical body in a Newtonian fluid in the Stokes' regime. The slender body propels itself by generating surface travelling waves. The mathematical framework to study this problem has been built using the slender-body theory of Keller and Rubinow (J. Fluid Mech., vol. 75, part 4, pp. 705-714, 1976). The motion of the body and the dilation of its surface are incorporated by having Stokeslet and source distributions along the body's axis, and the propulsion speed is determined by solving the resulting integral equation using an asymptotic expansion. For high wavenumbers, the propulsion speed for a cylinder is found to agree with results presented by Setter et al. (Phys. Rev. E 85, 066304, 2012) in the limit of vanishing cylinder radius and wave amplitude. We quantify the efficiency of the swimmer and explore its internal mechanics when the body is treated as a fluid-filled cavity with elastic fibres driving surface deformation.

2:31PM R17.00008 Hydrodynamic Contributions to Amoeboid Cell Motility , OWEN LEWIS, ROBERT GUY, University of California, Davis — Understanding the methods by which cells move is a fundamental problem in modern biology. Recent evidence has shown that the fluid dynamics of cytoplasm can play a vital role in cellular motility. The slime mold Physarum polycephalum provides an excellent model organism for the study of amoeboid motion. In this research, we use a simply analytic model in conjuction with computational experiments to investigate intracellular fluid flow in a simple model of Physarum. Of particlar interest are stresses generated by cytoplasmic flow which may be used to aid in cellular motility. In our numerical model, the Immersed Boundary Method is used to account for such stresses. We investigate the relationship between contraction waves, flow waves, adhesion, and locomotive forces in an attempt to characterize conditions necessary to generate directed motion.

2:44PM R17.00009 Gyrotactic Bioconvection in Density-Stratified Fluids[1] , ALIREZA KARIMI, AREZOO ARDEKANI, University of Notre Dame — Bioconvection is a complex phenomenon causing spontaneous pattern formation in some biological systems, for instance in bacterial, algal, and ciliate cultures. It occurs as a result of the collective behavior of up-swimming microorganisms in response to the certain types of physical stimuli. We are interested in the special case of gyrotaxis where the swimming is directed by the balance of the viscous torque arising from shear flow and the torque due to gravity acting on a bottom-heavy cell. We investigate gyrotactic bioconvection in the presence of a temperature or salinity stratification, using large-scale numerical simulations of a continuum model consisting of Navier-Stokes equations with Boussinesq approximation coupled with two conservation equations for the concentration of microorganisms and stratified agent. We explore different regimes of the flow by varying the corresponding dimensionless variables such as Rayleigh number and Lewis number, to shed light on the characteristics of double-diffusive convection engendered by active swimmers in a stratified environment.

[1]NSF Award CBET-1150348-CAREER

2:57PM R17.00010 Asymmetry and stability of shape kinematics in microswimmers' motion , YIZHAR OR, Faculty of Mechanical Engineering, Technion, Israel — Many swimming microorganisms governed by low-Reynolds-number hydrodynamics utilize flagellar undulations for self propulsion. Most of existing theoretical models assume that the shape kinematics is directly controlled, while in reality, eukaryotes actuate internal bending moments along their flagellum. Under control of the internal forces and torques, the swimmer's shape is dynamically evolving and periodic gaits may become unstable. In this work the problem is addressed by revisiting Purcell's three-link swimmer model where the joint torques are controlled. The swimmer's dynamic equations of motion are formulated and the underlying geometric symmetries are analyzed. It is found that certain symmetry properties of the input induce a reversing symmetry on the dynamics of the joint angles, under which periodic solutions are marginally stable. Moreover, it is proven that one has to break the front-back symmetry of the swimmer's structure and/or actuation profile in order to induce time-periodic solution for the shape kinematics which is asymptotically stable under perturbations. In particular, a swimmer with large drag resistance at its front enjoys a strongly stable shape kinematics. The results may explain why most of the flagellated eukaryotes swim with their head forward.

Tuesday, November 20, 2012 1:00PM - 2:57PM –

Session R18 Particle Laden Flows VII: 28D - Chair: Joe Barranco, San Francisco State University

1:00PM R18.00001 Dust Settling in Protoplanetary Disks with a Terminal Velocity Approach[1] , JOSEPH BARRANCO, DIANA MADERA, San Francisco State University — It is a remarkable fact that planets start out as microscopic grains within protoplanetary disks of gas and dust in orbit around newly-formed protostars, somehow growing by a factor of 10^{40} in mass in a period no more than 10^7 years. In the early stages of the planet formation, small dust grains settle into the midplane of the disk in a few thousand years. As the dust layer gets thinner, a vertical shear develops between the dust-rich layer at the midplane and the dust-poor gas above and below. Of great interest is whether such a layer will be unstable to Kelvin-Helmholtz instability (KHI), which will remix the dust with the gas, thwarting the formation of planets. In our previous work, we worked in the single-fluid limit in which the local dust-to gas ratio was an advectively conserved quantity (valid when the dust-gas friction time is very short). Here, we present new simulation in which this assumption is relaxed with a terminal velocity approach that allows dust to slowly drift apart from the gas. We investigate under what conditions dust may settle into dense layers at the midplane or get concentrated inside coherent vortices. Can dust concentrate enough to trigger a gravitational instability and clump up to form planetesimals, the building blocks of planets?

[1]Supported by NSF Astronomy & Astrophysics Research Grant.

1:13PM R18.00002 ABSTRACT WITHDRAWN –

1:26PM R18.00003 Numerical and experimental approaches to simulate soil clogging in porous media[1] , YULIYA KANARSKA, Lawrence Livermore National Laboratory, LLNL TEAM — Failure of a dam by erosion ranks among the most serious accidents in civil engineering. The best way to prevent internal erosion is using adequate granular filters in the transition areas where important hydraulic gradients can appear. In case of cracking and erosion, if the filter is capable of retaining the eroded particles, the crack will seal and the dam safety will be ensured. A finite element numerical solution of the Navier-Stokes equations for fluid flow together with Lagrange multiplier technique for solid particles was applied to the simulation of soil filtration. The numerical approach was validated through comparison of numerical simulations with the experimental results of base soil particle clogging in the filter layers performed at ERDC. The numerical simulation correctly predicted flow and pressure decay due to particle clogging. The base soil particle distribution was almost identical to those measured in the laboratory experiment. To get more precise understanding of the soil transport in granular filters we investigated sensitivity of particle clogging mechanisms to various aspects such as particle size ration, the amplitude of hydraulic gradient, particle concentration and contact properties. By averaging the results derived from the grain-scale simulations, we investigated how those factors affect the semi-empirical multiphase model parameters in the large-scale simulation tool.

[1]The Department of Homeland Security Science and Technology Directorate provided funding for this research

1:39PM R18.00004 Modeling Particle-Laden Compressible Flows Using Lattice-Boltzmann Simulation , ORLANDO AYALA, University of Delaware, JOHN THOMAS, Johns Hopkins University, LIAN-PING WANG, University of Delaware — Combing the three-dimensional compressible Lattice-Boltzmann model with a fluid-particle interaction model, a robust computational framework for predicting fluid/solid momentum transfer within a particle-laden compressible flow field is developed. This tool is used to examine the effects of a moving shock front on the time-evolution of the displacement, velocity, and acceleration vectors of a single spherical particle initially at rest. These results are compared to analytical solutions obtained from the Navier-Stokes equations for compressible flows and a relationship between Mach number and drag coefficient is developed. Next, momentum transport through a system of particles is examined and the effects of particle-particle interactions on shock front propagation/attenuation are discussed. These results are used to understand fluid/solid momentum transfer within detonation shock waves.

1:52PM R18.00005 Equation of Motion for a Drop or Bubble in Viscous Compressible Flows , M. PARMAR, S. BALACHANDAR, A. HASELBACHER, University of Florida — Here we theoretically analyze the unsteady motion of a bubble/drop with the inclusion of compressibility effects,but in the limit of vanishing Mach and Reynolds numbers. Linearized viscous compressible Navier-Stokes equations are solved inside and outside of the spherical bubble/drop and an expression of the transient force is first obtained in the Laplace domain and then transformed to the time domain. The total force is separated into the quasi-steady, the inviscid-unsteady, and the viscous-unsteady contributions. The quasi-steady and inviscid unsteady forces are the same as those given in the literature for an incompressible drop in an incompressible flow and for a particle in a compressible flow, respectively. For large times, the viscous unsteady force on a drop is the same as that in an incompressible flow. For acoustically short times, the viscous unsteady force is modified due to compressibility effects. Contrary to the finding in incompressible flow, where the viscous unsteady force on a bubble becomes non-singular for short times, the compressibility re-introduces an inverse-square-root decay rate. The present theoretical results are valid only in the limit of zero Mach- and Reynolds-numbers, using DNS we extend the results to finite Reynolds- and Mach-numbers.

2:05PM R18.00006 ABSTRACT MOVED TO M1.00010 –

2:18PM R18.00007 Particle-laden flow in a spiral separator , SUNGYON LEE, Dept. of Mathematics and Applied Mathematics Laboratory, University of California, Los Angeles, CA 90095, YVONNE STOKES, School of Mathematical Sciences, The University of Adelaide, South Australia, 5005, Australia, ANDREA BERTOZZI, Dept. of Mathematics and Applied Mathematics Laboratory, University of California, Los Angeles, CA 90095 — Spiral concentrators are used in the mining industry to separate particles of different size or density. The existing modeling literature considers the flow as a background fluid carrying non-neutrally buoyant particles. However recent work on modeling of slurries on inclines shows that at relatively modest volume fractions of particles, the presence of the particles affects the flow and, moreover, interparticle interactions such as hindered settling and shear-induced migration can quantitatively explain the dynamics of the separation of particle mixtures under gravity. We incorporate this physics into a model for particle segregation in a spiral concentrator.

2:31PM R18.00008 Bi-disperse particle-laden flows in the Stokes regime , GILBERTO URDANETA, UCLA, SARO MEGUERDIJIAN, USC, KALI ALLISON, Harvey Mudd, THOMAS CRAWFORD, University of Cambridge, WYLIE ROSENTHAL, Harvey Mudd, SUNGYON LEE, ALIKI MAVROMOUSTAKI, ANDREA BERTOZZI, UCLA — We present an experimental study which investigates the motion of bi-disperse suspensions consisting of ceramic (heavy) and glass (light) beads in PDMS oil flowing down an inclined plane under the action of gravity. Both types of beads are denser than the oil. We perform a parametric study in which we vary the inclination angle of the plane, the total particle volume fraction and the relative ratio of glass to ceramic beads. Mono-disperse suspensions of negatively buoyant particles give rise to three regimes: a "settled" regime in which particles settle to the substrate, a "ridged" regime in which particles settle to the front, and a transient, "well-mixed" regime in which settling does not occur. A similar trend is observed in the current study; lower inclination angles and higher concentrations of the ceramic beads favor the settled regime. Further, the addition of a second particle species induces a striking effect in which the heavier ceramic beads migrate on top of the lighter beads; this phenomenon is thought to be the result of competing forces in the direction normal to the flow arising from gravitational settling and shear-induced migration. We discuss the effect of experimental parameters on the location of the front versus time and changes in the fingering instability.

2:44PM R18.00009 Characterization of Oscillatory Boundary Layer Over a Closely Packed Bed of Sediment Particles[1] , JOSEPH SKITKA, SOURABH APTE, Oregon State University — Lack of accurate criteria for onset of incipient motion and sediment pickup function remain two of the biggest hurdles in developing better predictive models for sediment transport. To study pickup and transport of sediment, it is necessary to have a detailed knowledge of the small amplitude oscillatory flow over the sediment layer near the sea bed. Fully resolved direct numerical simulations are performed using fictitious domain approach (Apte et al., JCP 2009) to investigate the effect of a sinusoidally oscillating flow field over a rough wall made of regular hexagonal pack of spherical particles. The flow arrangement is similar to the experimental data of Keiller & Sleath (JFM 1976). Transitional and turbulent flows at $Re_\delta = 50, 100, 150, 200$ (based on the Stokes layer thickness, $delta$) are explored over a range of non-dimensional sphere sizes. The coherent vortex structures, turbulent cross-correlations and lift forces on the roughness elements are characterized for these flow conditions and compared against available data of Keiller & Sleath (JFM 1976) and Sleath (JFM 1986). The dynamics of the oscillatory flow over the sediment bed is used to understand the mechanism of sediment pick-up.

[1]Funding: NSF project #1133363, Sediment-Bed-Turbulence Coupling in Oscillatory Flows: Fully Resolved Numerical Experiments and Modeling

Tuesday, November 20, 2012 1:00PM - 3:10PM –

Session R19 Non-Newtonian Flows IV: Turbulence 28E - Chair: Yves Dubief, University of Vermont

1:00PM R19.00001 The onset of elasto-inertial turbulence , BJORN HOF, DEVRANJAN SAMANTA, Max Planck Institute for Dynamics and Self-Organization, YVES DUBIEF, School of Engineering, University of Vermont, MARKUS HOLZNER, Max Planck Institute for Dynamics and Self-Organization, CHRISTOF SCHAEFER, Saarland University, ALEXANDER MOROZOV, School of Physics & Astronomy, University of Edinburgh, CHRISTIAN WAGNER, Saarland University, JOSE MANUEL GALLARDO RUIZ, Max Planck Institute for Dynamics and Self-Organization — A new type of turbulence is discovered for elastic fluids such as dilute solutions of long chain polymers and surfactants. Experiments are carried out in channel and pipe flows with diameters ranging from a few centimeters to a few hundred microns. At large enough shear rates an instability is found giving rise to disordered motion. For sufficiently large concentrations this instability already occurs at very low Reynolds numbers, where for Newtonian fluids flows are always laminar. The data for different pipe diameters reveal that the onset of the instability is governed by the shear rate and not by the Reynolds number. The ensuing disordered flow has a larger drag than the laminar one. The friction scaling coincides with the well known maximum drag reduction asymptote inferring that this asymptote is the characteristic friction scaling of elasto-inertial turbulence.

1:13PM R19.00002 A new state of turbulence: Elasto-inertial turbulence[1] , YVES DUBIEF, University of Vermont, DEVRANJAN SAMANTA, MARKUS HOLZNER, Max Planck Institute for Dynamics and Self-Organization, Gottingen, Germany, CRISTOF SCHAFER, Saarland University, Saarbrucken, Germany, ALEXANDER MOROZOV, School of Physics & Astronomy, University of Edinburgh, UK, CHRISTIAN WAGNER, Saarland University, Saarbrucken, Germany, BJORN HOF, Max Planck Institute for Dynamics and Self-Organization, Gottingen, Germany, VINCENT TERRAPON, Aerospace & Mechanical Dept, University of Liege, Belgium, JULIO SORIA, Mechanical Engineering Dept, Monash University, Australia — The elasticity of polymer solutions is found to generate a new state of turbulence, elasto-inertial turbulence (EIT), characterized by an interplay between elastic and flow instabilities. Experiments and direct numerical simulations (DNS) in pipe and channel flows demonstrate the emergence of EIT at Reynolds numbers much lower than the critical Reynolds number for transition to turbulence in Newtonian flows. EIT causes the friction factor to deviate from the laminar solution and subsequently transition to the maximum drag reduction asymptote around Re=1800. EIT is a self-sustained mechanism that arises from the interactions between fluctuations of extensional viscosity, velocity and pressure. The polymer solution elasticity controls the growth of flow instability, resulting in transitional-like flows at high Reynolds numbers. The existence of EIT is not limited to pipe, channel or boundary layer flows, and evidence of EIT will be discussed in other flows, including natural convection using DNS.

[1]YD acknowledges the partial support of NIH grant No P01HL46703.

1:26PM R19.00003 Experimental measurements of turbulent polymer solutions[1] , ALEXANDRE DE CHAUMONT QUITRY, DOUGLAS H. KELLEY, NICHOLAS H. OUELLETTE, Yale University — Complex fluids modify turbulent flows over a broad range of scales, including scales much larger than the physical size of the fluid microstructure. Models have attempted to explain this phenomenon on the basis of assumptions such as enhanced effective viscosity or a critical length scale separating different flow regimes. We attempt to constrain such models with experimental measurements of bulk turbulence in a dilute solution of long-chain polyacrylamide in water. We use high-speed cameras and Lagrangian Particle Tracking Velocimetry to image the central region of a von Kármán swirling flow, in which counter-rotating impellers inject kinetic energy inertially. We observe that concentrations as low as a few parts per million can drastically modify the energy cascade.

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

1:39PM R19.00004 Nonlinear dynamics of turbulent drag reduction by polymers , MICHAEL GRAHAM, SUNG-NING WANG, University of Wisconsin-Madison, FRIEDEMANN HAHN, Univ. Stuttgart — Minimal channel flow of Newtonian and drag-reducing polymer solutions is studied computationally. Even in the Newtonian limit, intervals of "active" and "hibernating" turbulence exist, the latter displaying many features of the maximum drag reduction (MDR) asymptote observed in polymer solutions: weak streamwise vortices, nearly nonexistent streamwise variations and a mean velocity gradient that quantitatively matches experiments (i.e. the Virk log-law). Polymer stretching is very weak during hibernation. As viscoelasticity increases, the frequency of the hibernation intervals increases, leading to flows that increasingly resemble MDR. This observation can be explained with a simple mathematical model that posits that the lifetime of an active turbulence interval is the time that it takes for the turbulence to stretch polymer molecules to a certain threshold value beyond which the active turbulence is suppressed. An extended Karhunen-Loeve analysis is introduced and used to illustrate how the velocity and stress fields change as MDR is approached. These results and others indicate that the MDR dynamics are governed by an underlying Newtonian state – a saddle point in phase space – that is unmasked as viscoelasticity suppresses normal turbulent fluctuations.

1:52PM R19.00005 Elastic turbulence of polymer solutions at low Re in a straight channel , LAETITIA MARTINIE, JULIEN BEAUMONT, HUGUES BODIGUEL, Laboratoire du futur (UMR 5258), HAMID KELLAY, LOMA (UMR 5798), ANNIE COLIN, Laboratoire du futur (UMR 5258) — At low Reynolds number (Re<1), elastic turbulence develops in polymer solutions flowing in curvilinear flows for Weissenberg numbers Wi beyond a given threshold Wic. Unlike inertial turbulence (Re>>1), elastic turbulence is due to their normal stress anisotropy [Groisman and Steinberg, Nature, 2000]. It has only been shown very recently, both theoretically [Morozov and van Saarlos, Phys. Report, 2007] and experimentally [Bonn et al., PRE 2011], that elastic turbulence could also occur in rectilinear flows, provided that the perturbation amplitude is sufficiently high. In this work, we aim to characterize the consequences of this turbulence on the velocity profile and the flow rate-pressure relationship of high molecular weight Polyaccrylamide solutions flowing in a straight channel. By varying both flow rate and polymer concentration, we are able to explore a wide range of Wi. Flows driven by a controlled pressure in a microfluidic straight channel are characterized using particle image velocimetry. For Wi $<$ Wic, the measured velocity field is well accounted for by the bulk flow curve of the shear thinning fluid. For Wi $>$ Wic, this prediction is not valid anymore and a high level of fluctuations is observed. In addition, velocity profiles can be described by a logarithmic behavior in areas where the fluid is highly sheared, similarly to what is observed in inertial turbulence in a rectilinear geometry. A model based on an Olroyd B fluid behavior has been developed to explain the experimental profiles observed.

2:05PM R19.00006 Energy spectra in polymer-doped turbulent soap films[1] , RORY CERBUS, WALTER GOLDBURG, University of Pittsburgh, PINAKI CHAKRABORTY, OIST, NATHAN FLYNN, University of Pittsburgh, CHIEN-CHIA LIU, OIST — We investigate the energy spectra of turbulent soap films doped with a dilute amount of a very large molecular weight polymer ($>$ 1M). We perform experiments in a soap-film channel that in the absence of polymers manifests decaying turbulence and a direct enstrophy cascade: the energy spectrum E(k) $\propto$ k^{-3}. For polymer-doped flow, where the polymer is added gently to the soapy solution, we observe that the energy spectrum switches to E(k) $\propto$ k$^{-5/3}$, which is consistent with the inverse energy cascade of forced 2D turbulence. This switching of the spectral exponent from 3 to 5/3 occurs for polymer concentrations as low as 2 wppm. For lower concentrations, the spectral exponent is unaffected. We also find that our results are sensitive to the method of polymer doping. If we stir our polymer-doped solution repeatedly, the effect of the polymer diminishes: the exponent of the energy spectrum switches back from 5/3 to 3.

[1]Partially supported by OIST (Okinawa Institute of Science and Technology)

2:18PM R19.00007 Elastic Energy Transfer in Turbulence of Dilute Polymer Solution[1] , HENG-DONG XI, EBERHARD BODENSCHATZ, HAITAO XU, Max-Planck Institute for Dynamics and Self-Organization — We present an experimental study of the energy transfer in the bulk of a turbulent flow with small amount long-chain polymer additives. By varying the Reynolds numbers R_λ, Wissenberg number Wi and polymer concentration ϕ. We test quantitively the elastic theory proposed by de Gennes and Tabor (Europhys. Lett., 1986; Physica A, 1986). The rate of energy transfer by polymer elasticity as inferred from the theory is consistent with that measured from the second order Eulerian structure functions. The unknown parameter n in the theory, which represents the flow topology of the stretching field, is found to be nearly 1. Based on energy transfer rate balance, We propose an elastic length scale, r_ϵ, which describes the effect of polymer elasticity on turbulence energy cascade and captures the scale dependence of the elastic energy transfer rate.

[1]We are grateful to the Max Planck Society, the Alexander von Humboldt Foundation and the Deutsche Forschungsgemeinschaft for their support.

2:31PM R19.00008 Direct Numerical Simulation of Elastically Modified Turbulent Taylor-Couette Flow[1] , NANSHENG LIU[2], Dept.Chemical & Biomolecular Engineering, University of Tennessee, Knoxville, BAMIN KHOMAMI, Dept. Chemical & Biomolecular Engineering, University of Tennessee, Knoxville — Direct Numerical Simulations (DNS) of elastically modified turbulent Taylor-Couette (TC) flow are carried out to study the effect of polymer additives on the dynamics of the flow, using a fully spectral method in conjunction with the FENE-P model for the description of polymer chain dynamics. Significant polymer-induced drag increase is observed for the TC flow, which is strikingly different from the findings of drag reduction in the turbulent viscoelastic channel flow. Careful examination of turbulent, viscous and elastic stresses show that the elastically modified wall structures are mainly responsible for the polymer-induced drag increase. In addition, turbulence statistics are analyzed to develop the correlations between the polymer body force and velocity. The probability density functions (PDFs) of the velocity and polymer stress fluctuations are illustrated to reveal the stochastic characteristics of the flow.

[1]This work was supported by the NSF grant CBET-0755269 and NSFC grant NO. 10972211.
[2]Dept. Modern Mechanics, University of Science and Technology of China

2:44PM R19.00009 Mechanics and characteristics of transition to turbulence in elasto-inertial turbulence , VINCENT TERRAPON, Aerospace & Mechanical Engineering Dept, University of Liege, Belgium, JULIO SORIA, Mechanical Engineering Dept, Monash University, Australia, YVES DUBIEF, School of Engineering, University of Vermont VT — Numerical experiments of transition in elasto-inertial turbulent channel flows are used to highlight the mechanisms of transition and characterize the MDR regime. Specifically, the pressure kernel from the generalized pressure Poisson equation is used to demonstrate the role of elastic instabilities in inducing and sustaining a turbulent-like flow. Additionally, dynamic mode decomposition is applied to statistically steady viscoelastic flows at different Reynolds number to identify the relative contributions of elastic and inertial instabilities. It is shown that elastic instabilities can be triggered through long-range interactions from disturbances in the free-stream, similarly to by-pass transition, and are then sufficient to self-sustain. When the Reynolds number is increased, the relative contribution of inertial instabilities becomes more important, and the flow demonstrates features that are characteristic to Newtonian turbulent flows (e.g., streaks, quasi-streamwise vortices), although at lower intensity.

2:57PM R19.00010 Studying the Topology and Dynamics of Elasto-inertial Channel Flow Turbulence Using the Invariants of the Velocity Gradient Tensor and Dynamic Mode Decomposition[1] , JULIO SORIA, Mechanical Engineering Dept, Monash University, Australia, VINCENT TERRAPON, Aerospace & Mechanical Engineering Dept, University of Liege, Belgium, YVES DUBIEF, School of Engineering, University of Vermont, VT — Direct numerical simulations (DNS) of the transition to and fully developed elasto-inertial turbulence (EIT) of a polymer solution in a channel flow has been used as a basis for the study of the topology and dynamics of these flows. The Reynolds number in these DNS ranged from 500 to 5000. The topology of these flows was studied through the joint probability density functions (JPDFs) of the second and third invariants of the velocity gradient tensor (VGT), Q_A and R_A respectively and the JPDFs of the second invariants of the rate-of-strain tensor and the rate-of-rotation tensor, Q_S and Q_W respectively. The results suggest that these transitional and fully developed EIT flows are predominantly made up of vortex sheets. Dynamic mode decomposition has been undertaken on the second invariant of the VGT, Q_A, which reveals that the most amplified mode is a two-dimensional structure located in the near-wall region. A "discontinuity" is observed close to the wall, which corresponds closely to the location of extrema of the mean polymer extension and is hypothesized to be a critical layer.

[1]Conducted during the 2012 CTR Summer Program

Tuesday, November 20, 2012 1:00PM - 3:10PM –

Session R20 Turbulent Boundary Layers XI: Nonequilibrium and Transition 30A - Chair: Ugo Piomelli, Queen039;s University

1:00PM R20.00001 Osborne Reynolds' pipe flow: Direct computation from laminar through bypass transition to fully-developed turbulence , XIAOHUA WU, Royal Military College of Canada, PARVIZ MOIN, Stanford University, RONALD ADRIAN, JON BALTZER, Arizona State University, JEAN-PIERRE HICKEY, Royal Military College of Canada — The most fundamental internal flow, smooth pipe from a slightly perturbed laminar inlet state continuously through bypass transition to fully-developed turbulence, has been computed using DNS over an axial domain length of 250 pipe radii. In the fully-developed turbulent region, mean and second-order turbulent statistics including the rate of viscous dissipation show excellent agreement with those obtained from an additional simulation using the conventional streamwise periodic boundary condition over an axial domain length of 30 pipe radii. Friction factor follows analytical solution prior to breakdown, and agrees with Moody's correlation after the completion of transition. During transition it exhibits an overshoot. Breakdown of the laminar pipe flow is characterized by the formation of large Lambda-shaped vortices pointing upstream, followed by their subsequent generation of small hairpin packets inclined towards the downstream direction.

1:13PM R20.00002 Effects of mean and fluctuating pressure gradients on turbulence in boundary layers[1] , PRANAV JOSHI, XIAOFENG LIU, JOSEPH KATZ, Johns Hopkins University — This study focuses on the effect of mean and fluctuating pressure gradients on boundary layer turbulence. The mean favorable pressure gradient (FPG) is imposed by a sink flow. The streamwise and wall-normal (x,y) components of the material acceleration (Du/Dt and Dv/Dt, respectively) are calculated from time resolved 2D PIV data, and integrated spatially to obtain the pressure distribution. The mean FPG prevents vortical structures from rising away from the wall, decreasing the Reynolds stresses in outer region. Large scale pressure fluctuation gradients involve three dimensional flow structures. In both, zero pressure gradient (ZPG) and FPG boundary layers, large scale fluctuating adverse pressure gradients (∂p'/∂x>0) are preferentially associated with sweeps, as fluid approaching the wall is decelerating. Consequently, the outward transport of small-scale turbulence is suppressed, and the near-wall enstrophy increases. Conversely, ejections, high wall-normal enstrophy flux, and viscous vorticity production occur mostly during ∂p'/∂x<0 as the fluid accelerates by moving away from the wall. The near-wall enstrophy flux peaks due to the inherent near wall 3D structures when ∂p'/∂x<0 and u'>0.

[1]Sponsored by NSF

1:26PM R20.00003 Investigation of the Spreading Mechanism of Turbulent Wedges and Spots[1] , JEFF CHU, DAVID GOLDSTEIN, University of Texas at Austin — We investigate the physics of turbulent wedge and turbulent spot spreading in a nominally zero pressure gradient 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. We examined both instantaneous as well as time averaged turbulent wedge flow. We find that there are low speed streaks that remain stationary in time near the outer edge of the wedge. It is plausible that turbulent wedge spreading stems from a succession of such streaks due to instabilities introduced by the streak immediately preceding it upstream. The spreading mechanisms of turbulent spots are also investigated. Turbulent spots are artificially triggered and allowed to develop over physical and unphysical surfaces. Attempts are made to view both spot spreading and turbulent wedge spreading in one coherent picture.

[1]Supported by AFOSR grant FA 9550-08-1-0453

1:39PM R20.00004 Development of Turbulent Spots in Bypass Transition , KEVIN NOLAN, TAMER ZAKI, Imperial College London — The transition region in a boundary layer experiences sporadic bursts of localized turbulent spots. These spots spread as they are convected, and merge to sustain the turbulent boundary layer downstream. In this work, turbulent spots are identified and tracked from their point of inception in Direct Numerical Simulations (DNS) of bypass transition. The spreading angle, spatial extent and volume are recorded for each turbulent spot. The variation of these parameters is investigated for different pressure gradients. While the spreading angle depends on pressure gradient, the volumetric growth rate is found to be insensitive. The instantaneous structure of the spots is also examined in isolated events and in the ensemble average. The early stage of spot growth comprises a large-scale structure in the form of a streamwise-oriented vortex pair. The ensemble-averaged statistics are computed and demonstrate the important contributions to the turbulent-kinetic-energy budget within the structure of the turbulent patches.

1:52PM R20.00005 Measurement of entropy generation within bypass transitional flow , RICHARD SKIFTON[1], RALPH BUDWIG[2], University of Idaho, DONALD MCELIGOT, Idaho National Laboratory, JOHN CREPEAU, University of Idaho — A flat plate made from quartz was submersed in the Idaho National Laboratory's Matched Index of Refraction (MIR) flow facility. PIV was utilized to capture spatial vectors maps at near wall locations with five to ten points within the viscous sublayer. Entropy generation was calculated directly from measured velocity fluctuation derivatives. Two flows were studied: a zero pressure gradient and an adverse pressure gradient (β = -0.039). The free stream turbulence intensity to drive bypass transition ranged between 3% (near trailing edge) and 8% (near leading edge). The pointwise entropy generation rate will be utilized as a design parameter to systematically reduce losses. As a second observation, the pointwise entropy can be shown to predict the onset of transitional flow. This research was partially supported by the DOE EPSCOR program, grant DE-SC0004751 and by the Idaho National Laboratory.

[1]Center for Advanced Energy Studies
[2]Center for Ecohydraulics Research

2:05PM R20.00006 Simulations of equilibrium accelerating turbulent boundary layers over rough walls , JUNLIN YUAN, UGO PIOMELLI, Queen's University — Studies of favorable-pressure-gradient (FPG) turbulent boundary layer are important both for engineering and geophysical applications, and for the fundamental understanding of the inner-outer layer interactions in turbulent wall-bounded flows. The sink flow, a FPG flow that is statistically self-similar in the streamwise direction, removes the history-dependence of a developing flow and is a good test bed to perform quantitative comparison between roughness and FPG effects. We carry out direct and large-eddy simulations of rough-wall sink flows in the transitionally- and fully-rough regimes, with mild to medium acceleration. The results show that the Reynolds number and the friction coefficient are generally affected by FPG and the physical roughness height (blockage ratio), respectively. The mean flow and the roughness function weakly depend on the pressure gradient. Flow statistics show that center-line velocity and friction velocity absorb the effects of acceleration and roughness respectively; close to the wall, the roughness-FPG combined effects are mostly reflected in the total drag, with the wall-normal location of strong turbulence production noticeably affected by the roughness. Explanations are provided by an exam of the turbulence structures.

2:18PM R20.00007 Non-stationary boundary layers and energy dissipation in incompressible flows , MARIE FARGE, Ecole Normale Superieure, Paris, France, ROMAIN NGUYEN VAN YEN, Freie Universitaet, Berlin, Germany, KAI SCHNEIDER, Aix-Marseille University, Marseille, France — We argue that d'Alembert paradox (1749) is still unresolved for very large Reynolds number flows. Prandtl (1904) assumed that there exists a viscous boundary layer attached to the wall and predicted that the drag force dissipates energy there at a rate proportional to $Re^{-1/2}$. Kato (1984) proved that, in the limit of infinite Reynolds number, the energy dissipation rate tends to zero if and only if the solution of the Navier-Stokes equation converges towards the solution of the Euler equations (with the same initial data) and then occurs in a very thin boundary layer of thickness proportional to Re^{-1}. By performing direct numerical simulations of a dipole crashing into a wall we show that Kato's scaling is more appropriate than Prandtl's scaling as soon as the boundary layer detaches from the wall. Details can be found in Nguyen van yen, Farge and Schneider, PRL 106, 184502 (2011).

2:31PM R20.00008 Tow-tank investigation of the developing zero-pressure-gradient turbulent boundary layer , JUNG HOON LEE, YONG SEOK KWON, JASON MONTY, NICHOLAS HUTCHINS, The University of Melbourne — Experiments are conducted using image-based measurement techniques to analyse the development of a zero-pressure-gradient turbulent boundary layer from trip to a high Reynolds number state. The unique experimental facility consists of a 5m long plate towed through a 60 x 1.8 x 1.8 m tow tank at speeds of up to 3 m/s. Windows in the side of the tank permit high-speed image acquisition and particle image velocimetry as the plate passes by the static measurement system. The evolution of the boundary layer can then be analysed from inception to Reynolds numbers up to $Re_\tau = 6000$ (near the end of the plate). Here $Re_\tau = \delta U_\tau/\nu$ is the Kármán number where δ is boundary layer thickness, U_τ is wall-shear velocity, and ν is kinematic viscosity. The unique frame-of-reference for this experiment enables us to track coherent motions as they evolve in the developing boundary layer. An analysis of vortical motion associated with the spatially and temporally evolving boundary layer reveals the development of large-scale vortices that originate from the inner region and extend to the edge of the outer region. Furthermore, the lifetimes of such large-scale vortical events can be estimated.

2:44PM R20.00009 Onset of turbulent mean dynamics in boundary layer flow , CURTIS HAMMAN, TARANEH SAYADI, PARVIZ MOIN, Center for Turbulence Research, Stanford University — Statistical properties of turbulence in low Reynolds number boundary layers are compared. Certain properties are shown to approach an asymptotic state resembling higher Reynolds number flow much earlier during transition than previously thought. This incipient turbulence is less stochastic and more organized than developed turbulence farther downstream, but the mean dynamics and production mechanisms are remarkably similar. The onset of turbulence in our recent simulations is also similar to that observed in the bypass transition of Wu & Moin where continuous freestream turbulence, rather than small-amplitude linear waves, triggers transition. For these inflow disturbances, self-sustaining turbulence occurs rapidly after laminar flow breakdown without requiring a significant development length nor significant randomization. Slight disagreements with FST-induced bypass transition are observed that correlate with the extra strain a turbulent freestream would impose upon the near-wall dynamics. Nevertheless, the turbulence statistics are similar shortly after the skin-friction overshoot independent of upstream receptivity. This early onset of deterministic turbulence provides support for reduced-order modeling of turbulent boundary layers based on non-linear stability mechanisms.

2:57PM R20.00010 Low order oscillatory modeling of the inner layer of turbulent boundary layers[1] , PROMODE R. BANDYOPADHYAY, AREN M. HELLUM[2], Naval Undersea Warfare Center, Newport, RI — The visualization of the viscous sublayer (VSL) by Einstein & Li (1956) and others indicates an oscillatory character with varying periods of growth followed by Strouhal-like liquidation of spanwise vorticity into arrays of lifting hairpins. In streak PIV at 20 wall units due to Li, Adrian & Hanratty (1996), we notice a preponderance of dislocations. Therefore, we assume the sublayer to be in a permanent state of near-bifurcation irrespective of Reynolds number. To the lowest order, we model this process by Stuart-Landau (SL) oscillator equation. It is assumed that within a VSL cell, the oscillator is diffusively coupled along the span, the surface-normal growth is also diffusive—slowing as it thickens—and the outer layer provides the disturbance vector. The sublayer growth is followed by breakdown, creating a new outer layer disturbance vector for the next cycle. The SL equation is modified accounting for the above processes. The initial value solution of spanwise vorticity shows the development of nonuniformity, numerous dislocations and meandering streak-like structures that persist over extraordinarily large number of oscillatory cycles. Variation of the oscillator time scale shows the effects of increasing Reynolds number.

[1]Sponsor: ONR34
[2]Sponsor: ONR-ASEE Post-Doctoral Program

Tuesday, November 20, 2012 1:00PM - 2:44PM –

Session R21 General Fluids V 30B - Chair: Babak Shotorban, University of Alabama

1:00PM R21.00001 Spectral characteristics of atmospheric surface layer turbulence in Qatar , REZA SADR, ARINDAM SINGHA, Texas A&M University at Qatar, MICRO SCALE THERMOFLUIDS LABORATORY TEAM — Turbulent characteristics of atmospheric boundary layer are of utmost importance in modeling the large-scale meteorological processes, diffusion of atmospheric contaminants, heat transfer and evaporation from the earth surface. Meteorological data are available for some areas of the globe but are sparse in tropical regions. There had been some recent studies in tropical weather in southwestern Asia but no study is carried out in Persian Gulf region. The present study reports the micrometeorological data collected from an atmospheric measurement station in the coastal region of Doha, Qatar, to characterize the nature of atmosphere surface layer (ASL) and ocean wave in this region. In the present work turbulence velocity spectra in this region is presented and compared with the available data from other locations. Also, empirical relationship for the normalized dissipation function in this region is suggested. Finally, variation of different length scales with the stability parameter z/L is investigated and compare with the existing values in available literatures. This is the first ever study of ASL in this area, and is expected to be a foundation of further atmospheric research endeavors in Qatar.

1:13PM R21.00002 Spray Characterization of Gas-to-Liquid Synthetic Jet Fuels , KUMARAN KANNAIYAN, REZA SADR, Texas A&M University at Qatar, GTL JET FUEL CONSORTIUM TEAM — Gas-to-Liquid (GTL) Synthetic Paraffinic Kerosene (SPK) fuel obtained from Fischer-Tropsch synthesis has grabbed the global attention due to its cleaner combustion characteristics. GTL fuels are expected to meet the vital qualities such as atomization, combustion and emission characteristics of conventional jet fuels. It is imperative to understand fuel atomization in order to gain insights on the combustion and emission aspects of an alternative fuel. In this work spray characteristics of GTL-SPK, which could be used as a drop-in fuel in aircraft gas turbine engines, is studied. This work outlines the spray experimental facility, the methodology used and the results obtained using two SPK's with different chemical compositions. The spray characteristics, such as droplet size and distribution, are presented at three differential pressures across a simplex nozzle and compared with that of the conventional Jet A-1 fuel. Experimental results clearly show that although the chemical composition is significantly different between SPK's, the spray characteristics are not very different. This could be attributed to the minimal difference in fluid properties between the SPK's. Also, the spray characteristics of SPK's show close resemblance to the spray characteristics of Jet A-1 fuel.

1:26PM R21.00003 Optical properties of nanofluids and its implication in nPIV measurements , ANOOP KANJIRAKAT, REZA SADR, Texas A&M University at Qatar, MICRO SCALE THERMOFLUIDS LABORATORY TEAM — Nanofluids have shown potential as heat transfer fluids in recent times due to their anomalous enhancement in heat transfer characteristics. Optical experimental methods are used to study near-wall flow characteristics in nanofluids to better understand this phenomenon. It is important to characterize the optical properties of the fluid under consideration as accuracy of these measurement techniques highly depends on these characteristics. For example, evanescent wave based nano-Particle Image Velocimetry (nPIV) technique, that measures near-wall velocity fields with an out of plane resolution of O(100nm), is an effective tool for such studies In the present study, optical properties of SiO_2–water nanofluids at various particle concentrations are investigated. Measurements of refractive indices and the optical transmittance of nanofluids, which are directly related to the out-of-plane resolution of nPIV measurements, are reported. The effects of the modification of these optical properties on the nPIV measurements of nanofluids in a micro channel are then discussed.

1:39PM R21.00004 Compressible cavitation with stochastic field method , ANDREAS CLASS, JULIEN DUMOND, Karlsruhe Institut of Technology — Non-linear phenomena can often be well described using probability density functions (pdf) and pdf transport models. Traditionally the simulation of pdf transport requires Monte-Carlo codes based on Lagrange particles or prescribed pdf assumptions including binning techniques. Recently, in the field of combustion, a novel formulation called the stochastic field method solving pdf transport based on Euler fields has been proposed which eliminates the necessity to mix Euler and Lagrange techniques or prescribed pdf assumptions. In the present work, part of the PhD Design and analysis of a Passive Outflow Reducer relying on cavitation, a first application of the stochastic field method to multi-phase flow and in particular to cavitating flow is presented. The application considered is a nozzle subjected to high velocity flow so that sheet cavitation is observed near the nozzle surface in the divergent section. It is demonstrated that the stochastic field formulation captures the wide range of pdf shapes present at different locations. The method is compatible with finite-volume codes where all existing physical models available for Lagrange techniques, presumed pdf or binning methods can be easily extended to the stochastic field formulation.

1:52PM R21.00005 ABSTRACT WITHDRAWN –

2:05PM R21.00006 Surface manifestations of an underlaying turbulent flow , PABLO GUTIERREZ, SÉBASTIEN AUMAITRE, SPEC / CEA-Saclay — We present an experimental study of a turbulent flow in a quasi bidimensional configuration and with a free surface. Turbulence is excited in the volume of a liquid metal by using an electromagnetic forcing with a spatially tunable magnetic field. We will present our measurements of the velocity field at the surface, obtained by tracking particles, and of the surface deformation, obtained by a direct optical measurement. The turbulent flows under study show a strong correlation between the imposed forcing geometry and the mean velocity field. We also observed considerable deformation of the free surface and the preferential concentration of the particles used for visualization. Concequently, we will discusse possible physical scenarios at the origin of this concentration, which also depends on the forcing geometry.

2:18PM R21.00007 Dissolution without shrinking: a microfluidic study of multicomponent gas bubble dissolution , SUIN SHIM, Princeton University, JIANDI WAN, Rochester Institute of Technology, SASCHA HILGENFELDT, University of Illinois, HOWARD STONE, Princeton University — Spherical CO_2 bubbles generated in a flow-focusing microfluidic channel first shrink rapidly, and reach an equilibrium size. In the first – rapid dissolution – regime, the time needed for the bubbles to shrink in the channel is 30 ms regardless of surfactant concentrations in the liquid phase. After 30 ms following generation, all bubbles stop shrinking and reach an equilibrium radius, which varies with the surfactant concentration. We interpret the results by considering three major factors: interfacial tension, effects of other gases (O_2, N_2) that are already dissolved in the liquid phase, and the pressure drop along the channel. Our theoretical model, based on multicomponent 1D (radial) diffusion model, explains both shrinking and equilibrium regimes with different dissolution behavior of three gases and composition changes inside the bubble. We solve the model for a single gas bubble in an infinite liquid phase where the pressure changes with time, and compare with our experimental results.

2:31PM R21.00008 Generation and self-assembly of multiple droplets inside microchannels , JAN GUZOWSKI, PIOTR KORCZYK, Institute of Physical Chemistry, Polish Academy of Sciences, SLAWOMIR JAKIELA, PIOTR GARSTECKI, Institute of Physical Chemistry, Polish Academy of Sciences, MICROFLUIDICS AND COMPLEX FLUIDS GROUP TEAM — When two immiscible microscopic droplets immersed in a host fluid phase are brought into contact three different equilibrium topologies can form. Depending on the relative values of the three possible interfacial tensions one observes either complete engulfing of one drop by the other one, partial engulfing in which all three possible interfaces are present or non-engulfing when the drops remain separated by the host phase. In the case of multiple drops these surface interactions lead to self-assembly of complex stable and metastable architectures corresponding to local minima of the total interfacial energy. We study those equilibrium configurations experimentally by using automated microfluidic devices in which droplets are generated and merged on-demand inside microchannels. Guided by theoretical considerations and numerical energy minimization we propose stability diagrams spanned by ratios of surface tensions and volume fractions. We also explain how linear sequences of droplet volumes generated inside narrow microfluidic channels relax, after entering a wide chamber, towards given two- or three-dimensional architectures. Our findings provide routes towards synthesis of polymeric particles with predesigned internal structure and may find use in generation of autonomous droplet networks with application as biosensors.

Tuesday, November 20, 2012 1:00PM - 3:10PM –

Session R22 Turbulence Modeling 30C - Chair: Karan Venayagamoorthy, Colorado State University

1:00PM R22.00001 In Marriage of Model and Numerics, Glimpses of the Future[1] , ALIREZA NEJADMALAYERI, OLEG V. VASILYEV, ALEXEI VEZOLAINEN, University of Colorado Boulder — A newly defined concept of *m*-refinement (model-refinement), which provides two-way coupling of physical models and numerical methods, is employed to study the Reynolds scaling of SCALES with constant levels of fidelity. Within the context of wavelet-based methods, this new hybrid methodology provides a hierarchical space/time dynamically adaptive automatic smooth transition from resolving the Kolmogorov length-scale (WDNS) to decomposing deterministic-coherent/stochastic-incoherent modes (CVS) to capturing more/less energetic structures (SCALES). This variable fidelity turbulence modeling approach utilizes a unified single solver framework by means of a Lagrangian spatially varying thresholding technique. The fundamental findings of this computational complexity study are summarized as follows: 1) SCALES can achieve the objective of "controlling the captured flow-physics as desired" by profoundly small number of spatial modes; 2) Reynolds scaling of constant-dissipation SCALES is the same regardless of fidelity of the simulations; 3) the number of energy containing structures at a fixed level of resolved turbulent kinetic energy scales linearly with *Re*; and 4) the fractal dimension of coherent energy containing structures is close to unity.

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

1:13PM R22.00002 Detached-Eddy Simulation Based on the v^2-f Model[1] , SOLKEUN JEE, KARIM SHARIFF, NASA Ames Research Center — Detached-eddy simulation (DES) based on the v^2-f Reynolds-averaged Navier-Stokes (RANS) model is developed and tested. The v^2-f model incorporates anisotropy of near-wall turbulence, which is absent in other RANS models commonly used in the DES community. Here, we present preliminary but encouraging results for the proposed model. The constant, C_{DES}, required in the DES formulation was calibrated by simulating both decaying and statistically-stationary isotropic turbulence. Both cases provide the same value of C_{DES}, indicating that the forced case is an alternative way to determine the coefficient. After C_{DES} is calibrated, the v^2-f DES formulation is tested for flow around a circular cylinder at a Reynolds number of 3900, in which case turbulence develops after separation. Simulations indicate that this model represents the turbulent wake nearly as accurately as the dynamic Smagorinsky model. For comparison, Spalart-Allmaras-based DES is also included in the cylinder flow simulation.

[1]This research was supported by an appointment to the NASA Postdoctoral Program at the NASA Ames Research Center, administered by Oak Ridge Associated Universities through a contract with NASA.

1:26PM R22.00003 Finite-dimensional Asymptotics and Degrees-of-Freedom Estimation for Turbulence Models Incorporating Spectral Subgrid-scale Viscosity , JOEL AVRIN[1], University of North Carolina at Charlotte — We study the finite-dimensional large-time behavior of three-dimensional forced turbulence as modeled by a modified Navier-Stokes equation. Subgrid-scale viscous effects are modeled by adding a hyperviscous term, but only to the high frequencies past a cutoff wavenumber m. We theoretically establish for arbitrarily large Reynolds numbers that the asymptotic (i.e. large-time) behavior of the system is finite-dimensional with an estimate on the number of degrees of freedom well within the Landau-Lifschitz estimates. We also verify in the case that m is large enough that the overall large-time dynamics are controlled by the large-time dynamics of the inertial range. Given these promising results, we now would like to explore the physicality of the model by modifying the arguments underlying the Chapman-Enskog expansion.

[1]Department: Mathematics and Statistics

1:39PM R22.00004 Invariant turbulence models , ALEXANDER BIHLO, Centre de recherches mathématiques, Université de Montréal, ELSA MARIA DOS SANTOS CARDOSO-BIHLO, Faculty of Mathematics, University of Vienna, JEAN-CHRISTOPHE NAVE, Department of Mathematics and Statistics, McGill University, ROMAN POPOVYCH, Institute of Mathematics of NAS of Ukraine — Various subgrid-scale closure models break the invariance of the Euler or Navier–Stokes equations and thus violate the geometric structure of these equations. A method is shown which allows one to systematically derive invariant turbulence models starting from non-invariant turbulence models and thus to correct artificial symmetry-breaking. The method is illustrated by finding invariant hyperdiffusion schemes to be applied in the two-dimensional turbulence problem.

1:52PM R22.00005 Kolmogorov hypotheses for variable-resolution turbulence simulations , DASIA REYES, SHARATH GIRIMAJI, Texas A&M University — Variable-resolution (VR) turbulence computation approaches such as detached-eddy simulations (DES), hybrid RANS-LES, partially-averaged Navier-Stokes (PANS) methods and partially-integrated turbulence model (PITM) are gaining popularity in engineering applications. Justifiably, these methods can be considered direct numerical simulations (DNS) of a variable-viscosity (non-Newtonian) fluid. Subject to this paradigm, we extend Kolmogorov's first and second similarity hypotheses for VR calculations. The resulting scaling laws can be invaluable in assessing the physical validity of spatio-temporal fluctuations of VR methods. Investigation of PANS decaying isotropic turbulence shows that the resolved field Kolmogorov scales vary with resolution as expected.

2:05PM R22.00006 Differential filtering on unstructured grids with application to grid adaptation[1] , SANJEEB BOSE, PARVIZ MOIN, Center for Turbulence Research, Stanford University, FRANK HAM, Cascade Technologies Inc — Extension of explicitly filtered LES methods and their corresponding SGS models require a filtering operator that is low-pass on arbitrary meshes and can be decoupled from the underlying grid topology. Previously, we have utilized the differential filters proposed by Germano (1986) to perform explicitly filtered LES on unstructured grids. This framework is now extended to extract an estimate of the mean SGS kinetic energy to determine regions where mesh refinement is required. This procedure is automated using a local, anisotropic mesh refinement tool, adapt. This approach has been applied to large eddy simulation of a three-dimensional diffuser at Re=50,000, experimentally characterized by Kolade (2010). Results from two different mesh resolutions will be presented; an initially coarse mesh and a mesh refined using the SGS kinetic energy estimates. The adapted mesh has increased resolution in the separated shear layers originating from the bottom and side expanding walls. The accuracy of the SGS model will also be assessed through comparison of the LES predictions with experimental measurements. Other recent applications to flow over a cylinder with heat transfer and to flow over a turbine blade will be presented.

[1]Work has been supported by the DoE CSGF (DE-FG02-97ER25308) and the DoE PSAAP Program

2:18PM R22.00007 ABSTRACT WITHDRAWN –

2:31PM R22.00008 ABSTRACT MOVED TO R20.00010 –

2:44PM R22.00009 Computational modeling of scalar transport and buoyancy effects in turbulent flows using ODTLES , ALAN KERSTEIN, Consultant, CHRISTOPH GLAWE, HEIKO SCHMIDT, BTU Cottbus, RUPERT KLEIN, Freie Universitat Berlin, ESTEBAN GONZALEZ-JUEZ, Combustion Science and Engineering, Inc., RODNEY SCHMIDT, Sandia National Laboratories — ODTLES is a stochastic model for turbulent flow simulation consisting of a lattice-work of instantiations of the one-dimensional-turbulence (ODT) model, each of which is time advanced on a 1D domain with full spatial and temporal resolution. Collectively they form a 3D coarse mesh on which 3D flow is captured by coupling the 1D domains so as to obtain a formulation that reduces to direct numerical simulation (DNS) and conventional large-eddy simulation in the appropriate limits. The advantage of ODTLES relative to the latter is the built-in resolution of small scales where needed (near walls, across buoyancy jumps, etc.) at lower cost than resolving them using 3D DNS. A recent formulation targeting confined flow [1] is generalized to incorporate scalar fields and buoyancy effects. The generalized formulation, illustrative applications, and planned future development are described.

[1] E. D. Gonzalez-Juez, R. C. Schmidt, A. R. Kerstein, Phys. Fluids **23**, 125102 (2011).

2:57PM R22.00010 Commutative Recursive Filters for Explicit-Filter Large-Eddy Simulation of Turbulent Flows[1] , MYEONGKYUN KIM, DAEGEUN YOON, DONGHYUN YOU, Pohang University of Science and Technology — One of the most notable drawbacks associated with the implicit-filter LES is that the simulation result is dependent on the numerical grid employed due to the inherent dependence of the filtering operation on the numerical discretization. Alternatively, commutative explicit filters can be applied to distinguish the filtering operation from the underlying mesh distribution, thereby eliminating grid sensitivities. The efficacy of explicit-filtering to obtain grid-independent solutions of turbulent flows has been successfully demonstrated in the previous research (Bose, Moin & You, Phys. Fluids, 2010; Singh, You & Bose, Phys. Fluids, 2012). However, the use of broad-width filters accompanies significant increase in computational cost in terms of memory space and communication load for a distributed memory (MPI-based) parallel computation. To overcome the difficulty, a recursive filtering algorithm which can effectively replace a broad-width commutative filter with a series of narrow-width filters. The efficacy of the commutative recursive filtering method is evaluated in explicit-filter LES of turbulent channel flow, with particular attention to the performance of commutative recursive filters in terms of computational cost and memory requirement for a parallel computation.

[1]Supported by the National Research Foundation of Korea and the Brain Korea 21 Program of the Korea Research Foundation

Tuesday, November 20, 2012 1:00PM - 3:10PM –

Session R23 Turbulence Theory: Rotating/Stratified/Compressible 30D - Chair: Georgios Matheou, Jet Propulsion Laboratory

1:00PM R23.00001 Anisotropy statistics in homogeneous stratified sheared turbulence , GEORGIOS MATHEOU, Jet Propulsion Laboratory, California Institute of Technology, DANIEL CHUNG, Department of Mechanical Engineering, University of Melbourne — Stably stratified flows are prominent in many engineering and geophysical applications. Stratified turbulence is characterized by anisotropic large scales but for a high Reynolds number flow smaller scales are expected to become progressively more isotropic. We investigate the anisotropy characteristics of stationary homogeneous stratified sheared turbulence at various stratifications and Reynolds numbers. Three sets of direct numerical simulations are utilized with $Re_\lambda = 220$–800. For each Taylor Reynolds number, four simulations were carried out that range from neutral to very stably stratified conditions. Elementary anisotropy statistics are discussed and compared to estimates for the onset of local isotropy.

1:13PM R23.00002 Small-scale turbulence in stably stratified flows , SABA ALMALKIE, STEVE DE BRUYN KOPS , University of Massachusetts, Amherst — We study statistical characteristics of small-scale turbulence under the stabilizing effect of stratification using direct numerical simulations of horizontally homogeneous, vertically stratified turbulence. The simulations use up to $4096 \times 4096 \times 2048$ grid points to resolve the dissipation scales over a range of Froude and buoyancy Reynolds numbers. The focus is on the effects of large-scale anisotropy associated with different levels of stratification on the dynamics and isotropy of small scales. The isotropy of small scales is addressed in terms of full statistical analysis of the velocity gradient tensor up to the fourth order. Our results reveal the two dominant dynamics of stratified turbulence as three-dimensional turbulence and background stratified flow. These two competing dynamics affect each component of the velocity gradient tensor differently. As a result, statistical characteristics of kinetic energy dissipation rate depend on the stratification level. The probability density function of local energy dissipation rate reflects these two dominant dynamics by exhibiting a bimodal distribution. The results shed light on the definition of proper surrogates for energy dissipation rate in flows dominated by stratified turbulence.

1:26PM R23.00003 Acceleration Statistics in Rotating and Sheared Turbulence , FRANK JACOBITZ, University of San Diego, KAI SCHNEIDER, Aix-Marseille Université, WOUTER BOS, Ecole Centrale de Lyon, MARIE FARGE, Ecole Normale Supérieure — Acceleration statistics are of fundamental interest in turbulence ranging from theoretical questions to modeling of dispersion processes. Direct numerical simulations of sheared and rotating homogeneous turbulence are performed with different ratios of Coriolis parameter to shear rate. The statistics of Lagrangian and Eulerian acceleration are studied with a particular focus on the influence of the rotation 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 its 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.

1:39PM R23.00004 Scale locality and the inertial range in compressible turbulence , HUSSEIN ALUIE, Los Alamos National Laboratory — We use a coarse-graining approach to prove that inter-scale transfer of kinetic energy in compressible turbulence is dominated by local interactions. Locality here means that interactions between disparate scales decay at least as fast as a power-law function of the scale-disparity ratio. In particular, our results preclude transfer of kinetic energy from large-scales directly to dissipation scales, such as into shocks, in the limit of high Reynolds number turbulence as is commonly believed. The assumptions we make in our proofs on the scaling of velocity, pressure, and density structure functions are weak and enjoy compelling empirical support. Under a stronger assumption on pressure dilatation co-spectrum, we show that *mean* kinetic and internal energy budgets statistically decouple beyond a transitional "conversion" range. Our analysis demonstrates the existence of an ensuing inertial scale-range over which mean SGS kinetic energy flux becomes constant, independent of scale. Over this inertial range, mean kinetic energy cascades locally and in a conservative fashion, despite not being an invariant. We provide numerical support to our results on locality through an investigation of the cascade in the presence of shocks in Burger's flow.

1:52PM R23.00005 The role of helicity in stratified turbulence , CECILIA RORAI, DUANE ROSENBERG, ANNICK POUQUET, PABLO D. MININNI, National Center for Atmospheric Research (NCAR) — In magnetohydrodynamics (MHD) helicity plays an important role in the generation of large-scale magnetic fields; in atmospheric sciences, it has been claimed to be responsible for the stability of supercell thunderstorms, while in homogeneous and isotropic turbulence it is known to delay the energy decay but leave the statistical properties of the flow unaltered, thus being considered marginally relevant. However, recent numerical calculations have demonstrated that when rotation is introduced in the system, helicity plays an essential role. We report preliminary results on a numerical study of freely decaying strongly stratified turbulence, as occurs in the atmosphere and oceans, in the presence of helicity. The Boussinesq equations are integrated in a periodic domain with different initial conditions: a non-helical Taylor-Green flow, a fully helical Beltrami flow, and random flows with a tunable helicity. Different values of the Reynolds and Froude numbers are selected. The question we address is how these different initial velocity fields and helicity values affect the evolution of turbulence in terms of excitation of internal waves, energy decay and isotropic and anisotropic energy spectra.

2:05PM R23.00006 Modeling various effects of compressibility on the pressure Hessian tensor , SAWAN SUMAN, Indian Institute of Technology Delhi, SHARATH GIRIMAJI, Texas A&M University — Modeling the role of the pressure Hessian tensor in the evolution of turbulent velocity gradients is critical for developing closed Lagrangian equations of velocity gradients. In incompressible flows, substantial success has been achieved in this regard (Chevillard et al. Phys. Fluids, 2008). However, these incompressible models strongly hinge on Poisson equation of pressure, and thus - despite their success in incompressible flows - are not useful for compressible flows, wherein pressure behaves as a bona-fide thermodynamic variable evolving via the state and energy equations. Some initial attempts at modeling the pressure Hessian tensor inclusive of essential compressible physics have recently been made (Suman & Girimaji, J. Fluid Mech. 2009, 2011). However several further improvements are still desirable. With this motivation, we present a novel strategy of including further compressibility physics in these models by directly parameterizing a local state of the pressure Hessian tensor in terms of (i) local dilatation and (ii) rate of change of local dilatation. The rationale behind this modeling strategy and an evaluation of the model performance will be presented.

2:18PM R23.00007 Small-scale intermittency and shocks in high Reynolds number compressible turbulence[1] , DIEGO DONZIS, Texas A&M University — In many flows of interest turbulence interacts with shock waves. A canonical configuration is isotropic turbulence convected through a normal shock. Even without this shock, compressible flows develop so-called shocklets, which may affect the overall dynamics. It is also well-known that due to intermittency scales smaller than the mean Kolmogorov scale (associated with very large gradients) develop at high Reynolds numbers. It is, therefore, of interest to assess whether and under what conditions intermittent gradients can be comparable to those of shocks. Information about the most intense turbulence gradients is obtained from scaling exponents of structure functions. It is shown that in turbulence obeying Kolmogorov scaling, turbulence gradients become weak compared to shock gradients as Reynolds number increases. However, for turbulence with anomalous scaling, gradients are comparable to that of shocks. This provides a plausible mechanism for so-called broken regimes in shock-turbulence interactions where flow properties undergo smooth changes instead of a quasi-discontinuous jump across the shock. Furthermore, our DNS database is used to show that large gradients and velocities are correlated, an effect that increases the effectiveness of turbulence to disrupt shocks.

[1]Support from NSF and AFOSR is greatly acknowledged.

2:31PM R23.00008 A simple model for space-time correlation in compressible isotropic turbulence , DONG LI, LI GUO, XING ZHANG, GUOWEI HE, LNM, Institute of Mechanics, Chinese Academy of Sciences — Space-time correlation is fundamental to describe turbulent fluctuations in both space and time. Kraichnan proposes the sweeping model for space-time correlations in incompressible isotropic turbulence. Taylor's model for turbulent shear flows is broadly used although it is limited to the frozen-flow assumption. The extension of Taylor's model to non-frozen flows can be achieved by including the eddy distortion with experimental validation. However, these models don't apply to compressible turbulent flows. Lee et al (1992) develop a model for the compressible components of compressible flows. In this study, we will develop a model for space-time correlations of velocity fluctuations which contain both compressible and incompressible components. The model reveals two dynamic processes of turbulent fluctuations in compressible turbulence: (1) the compressible component propagates at the sound speed relative to local flow; (2) the local flow is convected by energy-contained eddies. The model is supported by direct numerical simulation of compressible isotropic turbulence in the sense of that all curves of the normalized time correlations for different wavenumbers collapse into a single one and their envelope is governed by the attenuation term in the present model.

2:44PM R23.00009 Statistics for One-dimensional Compressible Turbulence with Large-scale Forcing , QIONGLIN NI, SHIYI CHEN, SKLTCS, CAPT and HEDPS, College of Engineering, Peking University — A numerical study was performed to explore the difference between the one-dimensional hydrodynamic compressible turbulence and Burgers turbulence. The compressible flows were simulated at three different turbulent Mach numbers (M_t): 0.1, 1.0 and 3.2 using a large-scale random forcing scheme. We observed that the isentropic condition was approximately valid in the $M_t = 1.0$ case, and its statistical scalings were close to those in the Burgers equation. We then used the subensemble method to decompose the velocity field of the flow into two subensembles, according to the local energy fluxes in the positive and negative directions, respectively, and found that the subensemble probabilities were scale invariant in the inertial range. Further investigation revealed that the corresponding transition process between two subensembles in the compressible turbulence, unlike its Markovian counterpart in the Burgers turbulence, was not in accordance with a Markov process.

2:57PM R23.00010 On the cascade of kinetic energy in three-dimensional compressible turbulence , JIANCHUN WANG, YANTAO YANG, YIPENG SHI, ZUOLI XIAO, XIANTU HE, SHIYI CHEN, Peking University — A high resolution numerical simulation of three-dimensional compressible turbulence with large scale forcing is performed to study the kinetic energy transfer. In particular, the forcing scheme is designed to control the ratio of energy input from the solenoidal and compressive velocity components. Numerical simulation reveals that the compressive component of the density-weighted velocity has major contribution to the kinetic energy flux, due to the presence of large-scale shocks. Using a "coarse-graining" approach, we further show that the kinetic energy flux from both solenoidal and compressive components are nearly constant over the inertial range. However, the cascade rate of compressive mode is much faster than that of solenoidal mode, leading to the dominant of solenoidal kinetic energy over its compressive counterpart at high wavenumbers. We argue that this difference between the energy transfer rates is the major physical reason why the energy spectrum in the compressible turbulence always displays the Kolmogorov's -5/3 scaling in the inertial range, a phenomenon of incompressible turbulence.

Tuesday, November 20, 2012 1:00PM - 2:57PM –

Session R24 Compressible Flows II 30E - Chair: Joanna Austin, University of Illinois at Urbana-Champaign

1:00PM R24.00001 Planar Reflection of Detonations Waves[1] , JASON DAMAZO, JOSEPH SHEPHERD, California Institute of Technology — An experimental study examining normally reflected gaseous detonation waves is undertaken so that the physics of reflected detonations may be understood. Focused schlieren visualization is used to describe the boundary layer development behind the incident detonation wave and the nature of the reflected shock wave. Reflected shock wave bifurcation–which has received extensive study as it pertains to shock tube performance–is predicted by classical bifurcation theory, but is not observed in the present study for undiluted hydrogen–oxygen and ethylene–oxygen detonation waves. Pressure and thermocouple gauges are installed in the floor of the detonation tube so as to examine both the wall pressure and heat flux. From the pressure results, we observe an inconsistency between the measured reflected shock speed and the measured reflected shock strength with one dimensional flow predictions confirming earlier experiments performed in our laboratory.

[1]This research is sponsored by the DHS through the University of Rhode Island, Center of Excellence for Explosives Detection.

1:13PM R24.00002 Shock-boundary layer interaction and transonic flutter , PRADEEPA TUMKUR KARNICK, KARTIK VENKATRAMAN, Dept. of Aerospace Engineering, IISc, Bangalore — The transonic flutter dip of an aeroelastic system is primarily caused by compressibility of the flowing fluid. Viscous effects are not dominant in the pre-transonic dip region. In fact, an Euler solver can predict this flutter boundary with considerable accuracy. However with an increase in Mach number the shock moves towards the trailing edge causing shock induced separation. This shock-boundary layer interaction changes the flutter boundary in the transonic and post-transonic dip region significantly. We discuss the effect of viscosity in changing the flutter boundary in the post-transonic dip region using a RANS solver coupled to a two-degree of freedom model of the structural dynamics of a wing.

1:26PM R24.00003 Detailed Simulations and Analysis of Shock Bifurcation , YONG SUN, MATTHIAS IHME, Aerospace Engineering, University of Michigan, RALF DEITERDING, Oak Ridge National Laboratory — The interaction between the reflected shock wave and the boundary layer, that is formed by the incident shock, leads to shock bifurcation. This interaction induced inhomogeneities in the flow-field behind the reflected shock, and thereby affecting the combustion process in shock-tube ignition studies. To quantify effects of shock-bifurcation on the region behind the reflected shock, detailed simulations of a shock-tube system at high-pressure conditions are performed under consideration of detailed hydrogen reaction chemistry. Both 2D and 3D simulations are performed, and simulation results are compared against experiments and low-order shock-bifurcation models. To isolate relevant physical processes, additional simulations for different operating conditions, mixture-compositions, and adiabatic and isothermal walls are conducted, and results of this investigation are discussed in this presentation.

1:39PM R24.00004 Investigation of a transonic axisymmetric backward-facing step flow by means of time-resolved PIV and PSP , SVEN SCHARNOWSKI, MARTIN BITTER, CHRISTIAN J. KÄHLER, Bundeswehr University Munich — The results presented here are obtained within a sub project of the SFB TRR 40 program (founded by the German research foundation), which focuses on the analysis and modeling of coupled liquid rocket propulsion systems and their integration into the space transportation system. The overall objective is to develop technological foundations for the design of thermally and mechanically highly loaded components of future space transportation systems. The interaction between the shear layer and the nozzle in the wake of the launcher is particularly important. Therefore, detailed analyses of a generic space launcher model's wake flow are the main emphasis of this sub project. The combination of time-resolved PIV and PSP provide deep insights into the flow physics: The separation at the end of the main body, the formation of the shear layer, its growth and its reattachment, as well as the surface pressure fluctuations, are analyzed in detail. The results reveal unsteady loads caused by shear layer motion which could interfere with structural modes of a space launcher main engines' nozzle.

1:52PM R24.00005 Evolution of Imposed Vortices Over Concave Surfaces in Hypervelocity Flow , WILLIAM FLAHERTY, JOANNA AUSTIN, University of Illinois — Steamwise oriented vortices in the boundary layer of a hypersonic flow have the potential to affect heat transfer and skin friction significantly. These effects can be exacerbated by the addition of extra strain rates associated with concave surface curvature. Vortices can either occur naturally (in the form of Goertler vortices), or be introduced by some form of mechanical distortion (such as a protuberance). In this work we experimentally investigate the effect of concave surface curvature on the propagation of imposed vortices. These experiments are carried out in the Hypervelocity Expansion Tube at the University of Illinois. This facility is capable of generating flows with high enthalpies (4-9MJ/kg) and Mach numbers (3-7). Using a novel, fast-response pressure sensitive paint we are able to observe the development of vortices which are induced using diamond-shaped vortex generators. Models with varying amount of surface curvature (encompassing Goertler numbers between 10-22) are used to investigate the dynamics of vortex propagation and interaction. Our results show that the vortices remain attached and of constant strength for 10-12cm (80 boundary layer thicknesses) along the curved surfaces, while on flat plates the vortices are no longer apparent within 6 cm downstream.

2:05PM R24.00006 Shock focusing in water by convergent shell structures , CHUANXI WANG, VERONICA ELIASSON, University of Southern California — Lab scale experiments on shock focusing in water in convergent thin shell structures have been designed and performed. Thin shell structures made of two types of materials have been tested separately; carbon fiber and low carbon steel. The geometric shape of the structures is given by a logarithmic spiral, which is believed to maximize the amount of energy reaching the focal region from previous research. During an experiment, a shock wave in water is generated by projectile impact. High-speed schlieren photography is applied simultaneously to visualize the shock dynamics during the focusing event. Results show that the thin shell structure and the shock wave in the water are fully coupled and the interaction has some unique features, such as wave train patterns in water.

2:18PM R24.00007 ABSTRACT WITHDRAWN –

2:31PM R24.00008 Nitric oxide emission spectroscopy measurements in a hypervelocity post-shock flow field[1] , ANDREW SWANTEK, JOANNA AUSTIN, University of Illinois at Urbana Champaign — In hypervelocity flight conditions, typical of sub-orbital and reentry trajectories, the coupling between the fluid mechanics and the thermochemistry of the flow becomes important. In the current work, we use an expansion tube facility to accelerate air to hypervelocity test conditions (stagnation enthalpy 8MJ/kg, velocity 3.8 km/s). A double wedge model is used to generate an oblique shock, a strong bow shock, and a shock-boundary-layer interaction which is known to be very sensitive to the thermochemical state of the gas. We investigate the nitric oxide emission signal in the ultraviolet region (220-255 nm, A-X transition) at four spatial locations downstream of the bow shock (0, 2, 4, and 6 mm). An in-house code is used to simulate the spectrum in this region and thus obtain a temperature fit. Temperatures are observed to decrease when traversing downstream, starting at approximately the frozen temperature (about 7700 K) at the location of the shock (0 mm). The furthest downstream point deviates from this trend, potentially due to heating in a shear layer formed in the flow field. The flow field is seen to be in non-equilibrium in this region, as temperatures do not reach the equilibrium temperature (about 3900 K).

[1]This work was supported by an AFOSR award FA9550-11-1-0129 with Dr John Schmisseur as Program Manager.

2:44PM R24.00009 Shock Tube Investigation of Quasi-Steady Drag in Shock-Particle Interactions , JUSTIN WAGNER, STEVEN BERESH, SEAN KEARNEY, BRIAN PRUETT, ELTON WRIGHT, Sandia National Laboratories — A reassessment of historical drag coefficient data for spherical particles accelerated in shock-induced flows has motivated new shock tube experiments of particle response to the passage of a normal shock wave. Particle drag coefficients were measured by tracking the trajectories of 1-mm spheres in the flow induced by incident shocks at Mach numbers 1.68, 1.93, and 2.04, over test times of about 0.5 milliseconds. Previous shock tube studies conducted under similar test conditions have concluded that the unsteadiness associated with the accelerating particle resulted in elevated drag coefficients. However, recent theoretical work suggests that such effects should only last for microsecond timescales. Furthermore, low values for the acceleration parameter indicate that unsteadiness should be negligible. Consistent with past experiments, the current data clearly show that as the Mach number increases, the drag coefficient increases substantially. This increase significantly exceeds the drag predicted by incompressible standard drag models, but a recently developed compressible drag correlation returns values quite close to the current measurements. Consistent with recent theoretical work, these observations suggest that elevated particle drag coefficients are a quasi-steady phenomenon attributed to increased compressibility rather than true flow unsteadiness.

Tuesday, November 20, 2012 1:00PM - 2:31PM –

Session R25 Flow Control: General II 31A - Chair: Hamid Johari, California State University at Northridge

1:00PM R25.00001 Robustness of Input Shaping for Liquid Sloshing Suppression in a Horizontally Accelerating Container[1] , DONGJOO KIM, SEONG-WOOK HONG, KYOUNGJIN KIM, Kumoh National Institute of Technology — Input shaping has been recently shown to be effective in reducing liquid sloshing, which occurs when a partially filled container experiences acceleration for fast positioning control. However, its robustness to the change of system parameters needs to be further understood because input shaping is an open-loop control without feedback sensing. Therefore, the objective of this study is to investigate the robustness of input shaping as a means of suppressing liquid sloshing in a horizontally accelerating container numerically and experimentally. In the case of numerical simulation, the governing equations for unsteady two-phase fluid motions are solved rather than using the simple and common pendulum model for free surface motion. In this study, three different input shapers (ZV, ZVD, convolved ZV shapers) are considered and their sensitivity to system frequency variation is examined for various container accelerations. The control efficiency of input shapers is evaluated in terms of the amplitude of transient peak and residual oscillations of liquid sloshing. Detailed information including dynamic behaviors of free surface will be provided in the final presentation.

[1]Supported by Research Fund, Kumoh National Institute of Technology

1:13PM R25.00002 Estimation of the Concentration from a Gaseous Moving Source Using Collaborating Sensing Aerial Vehicles[1] , TATIANA EGOROVA, MICHAEL A. DEMETRIOU, NIKOLAOS A. GATSONIS, Worcester Polytechnic Institute — This work considers the estimation of the concentration field caused by a moving gaseous source. A model-based estimation scheme incorporates the vehicle dynamics in the estimation scheme in order to arrive at a guidance control law that is dictated by the performance of the estimator. The model-based estimation scheme provides on-line estimates of the concentration field and of the proximity of the moving source. The process state estimate is realized through the use of multiple sensing aerial vehicles (SAVs) that have collaborating capabilities. Each SAV implements its own model-based estimator using its own sensor measurements and shares its estimate with the remaining SAVs. The advantage of the collaborating scheme is the faster convergence of the process state estimate. Computational results demonstrate the advantage of the SAV collaboration in estimating the process state and the proximity of the moving source.

[1]Air Force Office of Scientific Research, Dynamics and Control Program, grant FA9550-12-1-0114

1:26PM R25.00003 Cavitation on Hydrofoils with Leading Edge Protuberances[1] , DERRICK CUSTODIO, CHARLES HENOCH, Naval Undersea Warfare Center (NUWC), Newport, RI, HAMID JOHARI, California State University, Northridge, OFFICE OF NAVAL RESEARCH COLLABORATION — The effects of spanwise-uniform sinusoidal leading edge protuberances on the flow characteristics and forces of finite-span hydrofoils under vaporous cavitation conditions were examined experimentally over angles of attack ranging from -9$^\circ \leq \alpha \leq 27^\circ$. Two planforms were studied, rectangular and swept, at a Reynolds number of $\approx$ 720,000. Two protuberance wavelengths, $\lambda = 0.25c$ and $0.50c$, and three amplitudes, $A = 0.025c$, $0.05c$, and $0.12c$, were examined as they resemble the humpback whale flipper morphology. All hydrofoils retain a mean NACA 63_4-021 profile. The forces and moments were measured at a freestream velocity of 7.2 m/s, and high-speed digital photography was used to capture flow field images at several angles of attack. The cavitation number corresponding to incipient leading edge cavitation was also calculated. As far as forces and cavitation number are concerned, results show that the baseline hydrofoil tends to have nearly equal or improved performance over the modified hydrofoils at most angles of attack tested. Flow images reveal that it is possible that the extent of sheet and tip vortex cavitation can be reduced with the introduction of leading edge protuberances. The forces and cavitation characteristics will be presented.

[1]Sponsored by the ONR-ULI program.

1:39PM R25.00004 An improved algorithm for balanced proper orthogonal decomposition using analytic tails[1] , JONATHAN TU, CLARENCE ROWLEY, Princeton University — Balanced proper orthogonal decomposition (BPOD) can be used in flow control applications to identify coherent structures of interest and to form reduced-order models. Doing so involves simulating impulse responses of the direct and adjoint systems, in order to compute factorizations of the empirical Gramians. We present a new variant of the BPOD algorithm that simultaneously reduces its computational cost and increases its accuracy. Dynamic mode decomposition (DMD) is used to identify the slow eigenvectors that dominate the long-time behavior of the impulse responses, and the contribution of these eigenvectors to the empirical Gramians is then accounted for analytically. This procedure greatly reduces the error inherent in truncating the impulse responses after a finite time. We demonstrate the effectiveness of this algorithm by applying it to the flow past a two-dimensional cylinder, at a Reynolds number of 100. Reduced-order models are computed for the restriction of the wake dynamics to the stable subspace. Models generated using the analytic tail method yield the same accuracy as those computed using traditional BPOD, with a 70% reduction in computation time.

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

1:52PM R25.00005 Large-Eddy Simulations of Plasma Control for Separated Supersonic Flow[1]

, NICHOLAS BISEK, JONATHAN POGGIE, Air Force Research Laboratory — The Navier-Stokes equations were solved using a high-fidelity time-implicit numerical scheme and an implicit large-eddy simulation approach to investigate plasma-based flow control for supersonic flow over a compression ramp. The configuration includes a flat-plate region to develop an equilibrium turbulent boundary-layer at Mach 2.25, which was validated against a set of experimental measurements. The fully turbulent boundary-layer flow traveled over a 24° ramp and produced an unsteady shock-induced separation. A control strategy to suppress the separation through a magnetically-driven gliding-arc actuator was explored. The size, strength, and placement of the actuator were developed based on recent experiments. Three control scenarios were examined: steady control, pulsing with a 50% duty cycle, and Joule heating. The results show the control mechanism reduced the time-mean separation length for all three situations. The case without pulsing and Joule heating was the most effective, with a reduction in the separation length by more than 75%. The controller was also found to significantly reduce the low-frequency content of the turbulent kinetic energy spectra within the separated region and reduce the total kinetic energy downstream of reattachment.

[1]Funded in part by the Air Force Office of Scientific Research, under a laboratory task monitored by Dr. J. Schmisseur, AFOSR/RSA. The computational resources were supported by a grant of supercomputer time from the U.S. Department of Defense.

2:05PM R25.00006 Control of Shock-Induced Boundary Layer Separation by using Pulsed Plasma Jets[1]

, BENTON R. GREENE, NOEL T. CLEMENS, The University of Texas at Austin, Austin, TX, DANIEL MICKA, Creare, Inc., Hannover, NH — Shock-induced turbulent boundary layer separation can have many detrimental effects in supersonic flow including flow instability, fatigue of structural panels, and unstart in supersonic inlets. Pulsed plasma jets (or "spark jets"), which are characterized by high bandwidth and the ability to direct momentum into the flow, are one promising method of reducing shock-induced separation. The current study is focused on investigating the efficacy of plasma jets to reduce the separated flow induced by a compression ramp in a Mach 3 flow. Three different 3-jet actuator configurations are tested: 20° pitched, 45° pitched, and 22° pitched and 45° skewed. The jets are pulsed at frequencies between 2 kHz and 4 kHz with duty cycles between 5 and 15%. The shock wave is generated using a 20° compression ramp, and the location of the shock-induced separation is visualized using surface oil streak visualization as well as particle image velocimetry. The results of the study show that of the three configurations, the plasma jets pitched at 20° from the streamwise direction cause the greatest reduction in separation, and when pulsed at a frequency of 3.2 kHz and 12% duty cycle can reduce the size of the separation region by up to 40%.

[1]This work is supported by AFRL under SBIR contract.

2:18PM R25.00007 4D-Var identification of POD Reduced-Order Models[1]

, LAURENT CORDIER, GILLES TISSOT, BERND R. NOACK, Institute PPRIME, France — A reduced-order modelling (ROM) strategy is crucial to achieve model-based control in a wide class of flow configurations. In turbulence, ROMs are mostly derived by Galerkin projection of first principles equations onto the proper orthogonal decomposition (POD) modes. These POD ROMs are known to be relatively fragile when used for control design. In this communication, a four-dimensional variational assimilation approach (4D-Var) is used to identify the coefficients of the POD ROM. Essentially, data assimilation combines imperfect observations, a background solution and the underlying dynamical principles governing the system under observation to determine an optimal estimation of the true state of the system. The methodology will be illustrated for a cylinder wake flow on two datasets of increasing dynamical complexity: i) a DNS at $Re = 200$, and ii) PIV measurements at about $Re = 40000$.

[1]Partially funded by the ANR Chair of Excellence TUCOROM and the ANR CORMORED

Tuesday, November 20, 2012 1:00PM - 3:23PM –

Session R26 Aerodynamics V 31B - Chair: Mingjun Wei, New Mexico State University

1:00PM R26.00001 Low Reynolds Number Wing Transients in Rotation and Translation[1]

, ANYA JONES, KRISTY SCHLUETER, University of Maryland — The unsteady aerodynamic forces and flow fields generated by a wing undergoing transient motions in both rotation and translation were investigated. An aspect ratio 2 flat plate wing at a 45 deg angle of attack was driven over 84 deg of rotation (3 chord-lengths of travel at 3/4 span) and 3 and 10 chord-lengths of translation in quiescent water at Reynolds numbers between 2,500 and 15,000. Flow visualization on the rotating wing revealed a leading edge vortex that lifted off of the wing surface, but remained in the vicinity of the wing for the duration of the wing stroke. A second spanwise vortex with strong axial flow was also observed. As the tip vortex grew, the leading edge vortex joined the tip vortex in a loop-like structure over the aft half of the wing. Near the leading edge, spanwise flow in the second vortex became entrained in the tip vortex near the corner of the wing. Unsteady force measurements revealed that lift coefficient increased through the constant-velocity portion of the wing stroke. Forces were compared for variations in wing acceleration and Reynolds number for both rotational and translational motions. The effect of tank blockage was investigated by repeating the experiments on multiple wings, varying the distance between the wing tip and tank wall.

[1]U.S. Air Force Research Laboratory, Summer Faculty Fellowship Program

1:13PM R26.00002 Unsteady Lift Response and Energy Extraction in Gusting Flows

, JEESOON CHOI, TIM COLONIUS, California Institute of Technology, DAVID WILLIAMS, Illinois Institute of Technology — The unsteady aerodynamic forces associated with streamwise (surging) and transverse (plunging) oscillating motions are studied to understand the dynamic response to gusts and the potential for energy extraction. We focus on 2D thin airfoils at low sub- and super-critical Reynolds number so that the role of wake instability can be isolated. Simulations are performed in a large parameter space of angle of attack, reduced frequency, and oscillation amplitude. At low angle of attack, the magnitude and phase of the fluctuating lift are in reasonable agreement with classical theory at all reduced frequencies. In this case, the quasi-steady force is modified by contributions from shed vorticity at the trailing edge and added-mass at high reduced frequency. At high angle of attack, the fluctuating forces are found to be enhanced or attenuated by a leading-edge vortex, depending on the reduced frequency. Resonance with the wake instability is also investigated.

1:26PM R26.00003 Transient Vortex Structures in the Near Wake of a Wing during Pitch Up/Down Maneuvers[1]

, EMILIO GRAFF, MORGANE GRIVEL, California Institute of Technology, DAVID WILLIAMS, Illinois Institute of Technology — The vorticity distribution in the wake of a thin airfoil reflects the lift and bound circulation history of the wing. During a pitch-up maneuver from 0 degrees to some higher angle of attack (assuming attached flow), a "starting vortex" is formed in the wake whose circulation is opposite in strength to the bound circulation in the wing. However, a finite time is required for the starting vortex to fully develop, and if the wing pitches down to a smaller angle of attack before the first starting vortex has reached full strength then an imbalance in the wake circulation occurs. The delay time between the up/down pitch motions and the maximum angle of attack determine which additional vortices must be formed to satisfy Kelvin's theorem. In addition to the irrotational flow vortices that form, vorticity associated with the viscous boundary layers also accumulates into discrete vortices that accompany each "starting vortex." The complicated distributions of vortices and their evolution in the wake are examined with detailed PIV, smoke-visualization, and numerical simulations at Re = 240 to 70,000.

[1]The support for David Williams by AFOSR Grant FA9550-09-1-0189 is gratefully acknowledged.

1:39PM R26.00004 Effects of wing flexibility on aerodynamic performance in hovering flight[1] , TAO YANG, MINGJUN WEI, New Mexico State University — In this study, we use a strong-coupling approach to simulate three dimensional flexible flapping wings in hovering flight. The approach is based on a uniform description of both fluid and solid in global Eulerian framework. There has been extensive validation of the current approach with other numerical simulation and experiments. Then we apply our approach to simulate flapping wings with different flexibility and other control parameters. The simulation results allow us to study directly the effects of wing flexibility on the aerodynamic performance of hovering flight.

[1]Supported by ARL

1:52PM R26.00005 Optimization of the airfoil stroke in a high Reynolds number flow for energy harvesting , XINJUN GUO, SHREYAS MANDRE, Brown University — We investigate the heaving and pitching stroke of an airfoil for maximum energy extraction from the flow of the ambient fluid. This analysis is targeted towards optimization of oscillating airfoil or hydrofoils for wind and hydrokinetic energy conversion respectively. The goal is to study the influence of unsteady aerodynamic effects like leading edge vortex, unsteady boundary layer separation, vortex recapture, etc. We are inspired by the mechanics of insect and bird flight, which are believed to use unsteady aerodynamics for enhanced performance. Our airfoil has two degrees of freedom, heaving and pitching, and these degrees of freedom are actuated independently. We employ a variational framework for optimizing the transient stroke of the airfoil with the objective function being the time-averaged harvested power.

2:05PM R26.00006 Experimental and Numerical investigations of flapping flight , SIDDHARTH KRITHIVASAN, SANTOSH ANSUMALI, SREENIVAS KR, JNCASR, Bangalore, EMU, JNCASR COLLABORATION — Insects have been observed to produce higher lift than predicted by conventional steady-aerodynamics using a combination of unsteady aerodynamic mechanisms. The wing kinematics and the flow fields produced during flapping flight is essentially 3D. Recently, in our group it has been shown, using flow visualization and 2D simulations, that the asymmetric flapping where, down-stroke is faster than the upstroke, can produce sustained lift. Also by introducing controlled wing flexibility, one can increase the magnitude of the lift. In order to verify these predictions quantitatively we are measuring forces produced by a mechanical flapper using a force-balance. Results of this study will be presented that includes the forces measured in symmetric and asymmetric flapping at different flapping frequencies. Similar understanding of various wing-kinematics during a forward flight can be achieved by doing transient, 3D simulations. A fast, accurate and simple 3D scheme which is capable of dealing with moving boundaries using Lattice Boltzmann has been developed for this purpose. Benchmarking of this scheme has been done for a forward flapping of the wing with elliptical cross-section. The results on the benchmarking and other preliminary results will be presented in the conference.

2:18PM R26.00007 The aerodynamic cost of flight in bats—comparing theory with measurement , RHEA VON BUSSE, RYE M. WALDMAN, SHARON M. SWARTZ, KENNETH S. BREUER, Brown University — Aerodynamic theory has long been used to predict the aerodynamic power required for animal flight. However, even though the actuator disk model does not account for the flapping motion of a wing, it is used for lack of any better model. The question remains: how close are these predictions to reality? We designed a study to compare predicted aerodynamic power to measured power from the kinetic energy contained in the wake shed behind a bat flying in a wind tunnel. A high-accuracy displaced light-sheet stereo PIV system was used in the Trefftz plane to capture the wake behind four bats flown over a range of flight speeds (1–6m/s). The total power in the wake was computed from the wake vorticity and these estimates were compared with the power predicted using Pennycuick's model for bird flight as well as estimates derived from measurements of the metabolic cost of flight, previously acquired from the same individuals.

2:31PM R26.00008 Flow Structure on a Flapping Wing: Quasi-Steady Limit[1] , CEM OZEN, Air Products and Chemicals Inc., DONALD ROCKWELL, Lehigh University — The three-dimensional flow structure on a rotating wing is determined using stereoscopic particle image velocimetry. The wing is a rectangular flat plate with an aspect ratio $AR = 2$; the effective angle of attack is $\alpha_{eff} = 45^{\circ}$ and the Reynolds number $Re = 15{,}150$. Emphasis is on comparison of the early stages of rotation with the late stage corresponding to the steady-state. The flow structure in the early stage involves a stable leading-edge vortex, and root, tip, and shed vortices. Along the span of the wing, the leading-edge vortex has pronounced concentrations of chordwise-oriented vorticity. These concentrations arise from the large-magnitude spanwise flow along the surface of the wing. At large angles of rotation, there is loss of the tip vortex, which is accompanied by loss of the chordwise-oriented vorticity due to eruption of the spanwise flow from the wing surface. In addition, patterns of downwash, spanwise velocity and spanwise vorticity flux are correlated with the local scale and degree of concentration of spanwise vorticity of the leading-edge vortex.

[1]This research program was funded by the Air Force Office of Scientific Research.

2:44PM R26.00009 Thrust Enhancement of Flapping Wings in Tandem and Biplane Configurations by Pure Plunging Motion , S. BANU YILMAZ, MEHMET SAHIN, M. FEVZI UNAL, Istanbul Technical University — The propulsion performance of flapping NACA0012 airfoils undergoing harmonic plunging motion in tandem and biplane wing configurations is investigated numerically. An unstructured finite volume solver based on Arbitrary Lagrangian-Eulerian formulation is utilized in order to solve the incompressible unsteady Navier-Stokes equations. Four different tandem and four different biplane wing combinations are considered. Various instantaneous and time-averaged aerodynamic parameters including lift and drag coefficients, vorticity contours and streamlines are calculated for each case and compared with each other. As a reference the single wing case corresponding to the deflected jet phenomenon in Jones and Platzer (*Exp. Fluids* 46:799-810, 2009) is also studied. In these simulations, the Reynolds number is chosen as 252, the reduced frequency of plunging motion ($k = 2\pi f/U_{\infty}$) is 12.3 and the plunge amplitude non-dimensionalized with respect to chord is 0.12. The solutions of the single wing case indicate dependence on the location of start-up vortices. Meanwhile the multiple wing configurations indicate that the highest thrust enhancement is obtained in one of the biplane cases where the two wings closely moving towards each other namely biplane asynchronous-closer case.

2:57PM R26.00010 Flight Stabilization with Flapping Wings in Gusty Environments , CHAO ZHANG, LINGXIAO ZHENG, The Johns Hopkins University, TYSON HEDRICK, The University of North Carolina, RAJAT MITTAL, The Johns Hopkins University, FSAG TEAM — Achieving stable flight with flapping wings, is one of the major challenges for designing micro- aerial- vehicles (MAVs) but is part of the natural behavior of flying insects. To better understand how flying insects flyers can stabilize themselves during hovering flight, we use a computational model, which couples the Navier-Stokes equations for the aerodynamics with a six-degree of freedom (NS-6-DOF) flight dynamics model to recreate the free hovering flight of a hawkmoth. The NS-6DOF model indicates that a hovering hawkmoth is open-loop unstable. Examination of the aerodynamic forces and flight dynamics coupled with observations of the animal in the laboratory suggest a bioinspired strategy for close-loop stabilization of the hovering hawkmoth and this strategy is explored using the NS-6DOF insect model. Simulations are conducted both for quiescent and highly "gusty" ambient conditions and the computed response of the "stabilized" animal compared to experimental observations.

3:10PM R26.00011 Adjoint-based optimization for flapping wings[1] , MIN XU, MINGJUN WEI, New Mexico State University — Adjoint-based methods show great potential in flow control and optimization of complex problems with high- or infinite-dimensional control space. It is attractive to solve an adjoint problem to understand the complex effects from multiple control parameters to a few performance indicators of the flight of birds or insects. However, the traditional approach to formulate the adjoint problem becomes either impossible or too complex when arbitrary moving boundary (e.g. flapping wings) and its perturbation is considered. Here, we use non-cylindrical calculus to define the perturbation. So that, a simple adjoint system can be derived directly in the inertial coordinate. The approach is first applied to the optimization of cylinder oscillation and later to flapping wings.

[1]Supported by AFOSR

Tuesday, November 20, 2012 1:00PM - 3:10PM –

Session R27 Computational Fluid Dynamics IX 31C - Chair: Onkar Sahni, Rensselaer Polytechnic Institute

1:00PM R27.00001 Non-isothermal 3D SDPD Simulations[1] , JUN YANG, RAFFAELE POTAMI, NIKOLAOS GATSONIS, Worcester Polytechnic Institute — The study of fluids at micro and nanoscale requires new modeling and computational approaches. Smooth Particle Dissipative Dynamics (SDPD) is a mesh-free method that provides a bridge between the continuum equations of hydrodynamics embedded in the Smooth Particle Hydrodynamics approach and the molecular nature embedded in the DPD approach. SDPD is thermodynamically consistent, does not rely on arbitrary coefficients for its thermostat, involves realistic transport coefficients, and includes fluctuation terms. SDPD is implemented in our work for arbitrary 3D geometries with a methodology to model solid wall boundary conditions. We present simulations for isothermal flows for verification of our approach. The entropy equation is implemented with a velocity-entropy Verlet integration algorithm Flows with heat transfer are simulated for verification of the SDPD. We present also the self-diffusion coefficient derived from SDPD simulations for gases and liquids. Results show the scale dependence of self-diffusion coefficient on SDPD particle size

[1]Computational Mathematics Program of the Air Force Office of Scientific Research under grant/contract number FA9550-06-1-0236

1:13PM R27.00002 Comparison study of meshfree methods for viscous flow , ZHENYU HE, LOUIS ROSSI, University of Delaware — We compare and contrast two meshfree schemes for viscous flow: Smoothed particle hydradynamics (SPH) and vortex methods (VM). SPH and VM are widely used meshfree particle in fluid dynamic applications. SPH is more flexible for capturing multiphysics problems. VM is better developed theoretically but has a more limited scope of applications. In SPH, the state of fluid system is represented by a set of moving basis functions which represent material properties such as density and momentum. Vortex particle methods represent a discretization of the vorticity field and use a Greens kernel to determine the velocity field. Our aim is to clarify the role played by the most commonly used viscous terms in SPH and VM in simulating incompressible fluid flow. Special test problems are used in order to remove the boundary effect to the results. We will present the accuracy and the efficiency of the different schemes which highlight the importance of key parameters in the algorithms including core width, overlap and equations of state.

1:26PM R27.00003 Scale-bridging schemes based on the material point method[1] , SHAOLIN MAO, XIA MA, VIRGINIE DUPONT, DUAN ZHANG, Los Alamos National Laboratory — With recent development of heterogeneous computational resources, such as combined GPU and CPU computations, there is an emerging possibility to apply a continuum approach to thermodynamically non-equilibrium systems with the closure quantities, such pressure, computed directly from numerical simulation of systems at a smaller length and time scales. Although it may not be possible to calculate the entire physical system at the small length and time scales, one can calculate closure quantities at representative locations, such as Gauss points in a finite element calculation, or mesh nodes, cell centers in a finite volume calculation, by surrounding those points with small volumes, called sub-systems, and perform directly numerical simulation on them to consider physical interactions at the smaller time and length scales. Before this hopeful method can be practical, we need to study issues related history dependency, time interval and spatial size of the sub-systems to simulate in order to calculate the closure quantities with credible accuracy. We also need to study methods to communicate between the sub-systems. For this purpose, we develop a numerical scheme base on the material point method (MPM). Results from such studies and from the numerical scheme will be presented.

[1]This work was performed under the auspices of the United States Department of Energy.

1:39PM R27.00004 Artificial Compressibility with Entropic Damping , JONATHAN CLAUSEN, SCOTT ROBERTS, Sandia National Laboratories — Artificial Compressibility (AC) methods relax the strict incompressibility constraint associated with the incompressible Navier–Stokes equations. Instead, they rely on an artificial equation of state relating pressure and density fluctuations through a numerical Mach number. Such methods are not new: the first AC methods date back to Chorin (1967). More recent applications can be found in the lattice-Boltzmann method, which is a kinetic/mesoscopic method that converges to an AC form of the Navier–Stokes equations. With computing hardware trending towards massively parallel architectures in order to achieve high computational throughput, AC style methods have become attractive due to their local information propagation and concomitant parallelizable algorithms. In this work, we examine a damped form of AC in the context of finite-difference and finite-element methods, with a focus on achieving time-accurate simulations. Also, we comment on the scalability of the various algorithms. Sandia National Laboratories is a multi-program laboratory managed and operated by Sandia Corporation, a wholly owned subsidiary of Lockheed Martin Corporation, for the U.S. Department of Energy's National Nuclear Security Administration under contract DE-AC04-94AL85000.

1:52PM R27.00005 Solving Navier-Stokes' equation using Castillo-Grone's mimetic difference operators on GPUs , MOHAMMAD ABOUALI, JOSE CASTILLO, Computational Science Research Center, San Diego State University — This paper discusses the performance and the accuracy of Castillo-Grone's (CG) mimetic difference operator in solving the Navier-Stokes' equation in order to simulate oceanic and atmospheric flows. The implementation is further adapted to harness the power of the many computing cores available on the Graphics Processing Units (GPUs) and the speedup is discussed.

2:05PM R27.00006 Mathematical modelling of backflushing in membrane separation , FRANK VINTHER, Technical University of Denmark — A mathematical model of pressure driven membrane separation is presented. Concentration polarization is well known to reduce the flux, and thereby the performance of the separation process, due to an increase in osmotic pressure. Therefore, back shocking where the pressure difference and the flow through the membrane is reversed, is known to decrease the concentration polarization and increase the performance of the membrane. The interplay between back shocking amplitude, frequency and reduction in concentration polarization at the membrane surface is investigated.

2:18PM R27.00007 Size-Dependent Couple-Stress Fluid Mechanics and Application to the Lid-Driven Square Cavity Flow , AREZOO HAJESFANDIARI, GARY DARGUSH, ALI HADJESFANDIARI, University at Buffalo, State University of New York — We consider a size-dependent fluid that possesses a characteristic material length l, which becomes increasingly important as the characteristic geometric dimension of the problem decreases. The term involving l in the modified Navier-Stokes equations

$$\rho\frac{D\mathbf{v}}{Dt} = -\nabla p + \mu\nabla^2\mathbf{v} - \mu l^2\nabla^2\nabla^2\mathbf{v}$$

generates a new mechanism for energy dissipation in the flow, which has stabilizing effects at high Reynolds numbers. Interestingly, the idea of adding a fourth order term has been introduced long ago in the form of an artificial dissipation term to stabilize numerical results in CFD methods. However, this additional dissipation has no physical basis for inclusion in the differential equations of motion and is never considered at the boundary nodes of the domain. On the other hand, our couple stress-related dissipation is physically motivated, resulting from the consistent application of energy principles, kinematics and boundary conditions. We should note, in particular, that the boundary conditions in the size-dependent theory must be modified from the classical case to include specification of either rotations or moment-tractions. In order to validate the approach, we focus on the lid-driven cavity problem.

2:31PM R27.00008 Fluctuating hydrodynamics for coarse-grained implicit solvent models of soft materials , PAUL ATZBERGER, University of California Santa Barbara — Many coarse-grained models have been developed and effectively employed in equilibrium studies of soft materials by treating implicitly interactions mediated by the solvent. These include models for polymeric materials, gels, and lipid bilayer membranes. However, dynamic studies require at a minimum incorporating momentum transfer through the solvent degrees of freedom. Hydrodynamic theories often give a sufficient kinetic description of the solvent mediated coupling of microstructures. In soft materials such theories must be reconciled with thermal fluctuations and entropic effects that play a significant role. In this talk we discuss new approaches based on fluctuating hydrodynamics to incorporate such features into implicit solvent coarse-grained models. For efficient simulations of the solvent-microstructure interactions and thermal fluctuations, we introduce new numerical methods based on the Stochastic Eulerian Lagrangian Method (SELM). We then discuss results for specific coarse-grained models of polymeric fluids, gels, and lipid bilayer membranes.

2:44PM R27.00009 Development of a Computational Tool for Inductively Coupled Plasma Flow Over Test Samples , MAXIMILIAN DOUGHERTY, DOUGLAS FLETCHER, University of Vermont — A boundary layer flow code is being developed to complement experimental work at the University of Vermont Plasma Diagnostics Laboratory. The stagnation region boundary layer is important because it controls the heat flux to the material during planetary entry and ground testing. Within the nonequilibrium boundary layer, highly exothermic chemical reactions can significantly augment heat transfer to the material surface. Many boundary layer codes rely on similarity solutions in transformed coordinate systems that are not necessarily intuitive at first glance. The benefit of these transformations is that a simplified grid can be used with a finite difference approach but this is at the expense of obfuscating the basic physical quantities which are of interest from an experimental perspective. As a result, the code being developed will use a finite volume formulation in physical coordinates. Chemical and thermodynamic properties will be computed with a separate gas property library which will allow for calculations of high temperature dissociated multi-species flows.

2:57PM R27.00010 Coalescence of two colliding liquid droplets with lattice Boltzmann method[1] , YIKUN WEI, YUEHONG QIAN, Institute of Applied Mathematics and Mechanics, Shanghai University — In this talk, we present two-dimensional numerical simulations of the head-on and off-center binary collision of van der Waals liquid droplets using the lattice Boltzmann method (LBM). The effects of the Weber number, impact velocity, droplet size ratio on the coalescence process are investigated. Numerical results are found in a good agreement with experimental findings.

[1] Y.H. Qian, D. d'Humieres, and P. Lallemand, Europhys. Lett. 17, 479 (1992).
[2] X.W. Shan and H.D. Chen, Lattice Boltzmann model for simulating flows with multiple phases and components, Physical Review E 47 (3) (1993) 1815–1819.
[3] M.R. Swift, W.R. Osborn, and J.M. Yeomans, Lattice Boltzmann simulation of nonideal fluids, Physical Review Letters 75 (5) (1995) 830–833.
[4] Y.K. Wei Y.H. Qian, Lattice Boltzmann Simulations for Multiphase Fluids with the Redich-Kwong Equation of State, Journal of Hydrodynamics, 2011,23(6)823-828

[1]The research is supported in part by Shanghai Science and Technology Commission Projects of (1) Leading Scientists and (2) No. 11XD1402300.

Tuesday, November 20, 2012 1:00PM - 2:44PM –

Session R28 General Fluids III 32A - Chair: Alfonso Ganan-Calvo, ESI, Universidad de Sevilla

1:00PM R28.00001 Thermodynamics of continuous media with permanent electric polarisation and magnetisation , SYLVAIN BRECHET, JEAN-PHILIPPE ANSERMET, Institute of Condensed Matter Physics, EPFL, 1015 Lausanne, Switzerland — The thermodynamics of an electrically charged, multicomponent fluid with permanent electric polarisation, permanent magnetisation and intrinsic vorticity is analysed in the presence of electromagnetic fields with magnetoelectric coupling in the classical limit. Three equations characterising the fluid are derived: a thermostatic equilibrium equation, a reversible and an irreversible thermodynamic evolution equation. These equations are obtained by taking into account the first and second laws of thermodynamics, the chemical reactions, the second law of Newton in translation and in rotation, the local time evolution of the permanent polarisation and the permanent magnetisation, and Maxwell's equations. Explicit expressions for the temperature and the chemical potentials are derived in terms of the electromagnetic fields, the permanent electric polarisation, the permanent magnetisation, the intrinsic vorticity and the magnetoelectric coupling. The analysis of the irreversible thermodynamics yields novel dissipative equations accounting in particular for dielectrophoresis, magnetophoresis, the relaxation of the permanent electric polarisation and the permanent magnetisation, and other properties of electrorheological and magnetorheological fluids.

1:13PM R28.00002 Physical symmetries of Taylor cone-jets: foundations of scaling laws[1] , ALFONSO M. GANAN-CALVO, ESI, Universidad de Sevilla, JOSE M. MONTANERO, NOELIA REBOLLO-MUNOZ, Universidad de Extremadura — In this work, we aim to establish the scaling laws for the liquid rate of flow naturally ejected by quasi-steady Taylor cone-jets. To this end, we utilize an ample literature in the field reporting precise measurements of the electric current transported and the resulting droplet size as a function of liquid properties and flow rate. The projection of thousands of experimental conditions onto an appropriate non-dimensional parameter space maps a region bounded by the minimum rate of flow attainable in steady state. In this limit, a theoretical model here proposed teaches that a remarkable system of symmetries rises at the geometrical transition from the cone to the jet. This system of symmetries determines an inescapable scaling for the minimum flow rate and related variables. If the flow rate is further decreased, those symmetries break down (the system bifurcates: global instability & dripping). Our model predicts the minimum flow rates reached in experiments reported so far in the literature, including all ranges of liquid properties. The existing literature and a set of new experiments performed for this specific purpose confirms our predictions.

[1]The Ministry of Science and Education (Spain) supported this work through Grant No. DPI2010-21103-C04.

1:26PM R28.00003 The Minimum Flow Rate Scaling in Taylor Cone-Jets , WILLIAM SCHEIDELER, CHUAN-HUA CHEN, Duke University — A minimum flow rate is required to maintain steady electrohydrodynamic Taylor cone-jets, and is empirically known to be inversely proportional to the electrical conductivity of the working fluid in the electrospraying literature. Here, we show that this scaling law governed by the charge relaxation process is only applicable to low-viscosity liquids. At higher viscosities, the minimum flow rate is governed by the capillary-inertial process and exhibits a strong dependence on the nozzle diameter instead of the liquid conductivity. The two scaling laws are demarcated by the Ohnesorge number based on the nozzle diameter.

1:39PM R28.00004 Stability of a viscous pinching thread , JENS EGGERS, University of Bristol, Department of Mathematics — We consider the dynamics of a fluid thread near pinch-off, in the limit that inertial effects can be neglected. There exists an infinite hierarchy of similarity solutions corresponding to pinch-off. Only one of the similarity solutions (the "ground state") is stable, all other solutions are linearly unstable to perturbations, and thus cannot be observed. Eigenvalues and eigenfunctions are calculated analytically.

1:52PM R28.00005 Laboratory experiments on stratified flow through a suspended porous fence , SARAH DELAVAN, University at Buffalo, The State University of New York, ROGER NOKES, University of Canterbury, DAVID PLEW, NIWA — This study explores stratified flow through a suspended, porous, fence-like obstacle to simulate flow through fish farm cages, mussel farm rope suspensions, flow through suspended aquatic vegetation, underwater energy production structures, or windbreak and wave break fencing. Laboratory experiments were performed in a density stratified, stationary flume with a suspended porous fence model using a particle tracking velocimetry (PTV) system. Experiments explored the effect on the fluid of the fence depth to total depth ratio, the system Richardson number, and the porosity of the fence. Preliminary results suggest that the density stratification of the fluid inhibits vertical fluid motion, that fence porosity greatly controls the vertical mixing of the fluid, and that there may be an optimal fence depth to total depth ratio for full development of the system flow structures.

2:05PM R28.00006 Solutions to separated and quasi-laminarized pressure gradient boundary layers via similarity analysis , RIKI MINORU HOPKINS, RAUL BAYOAN CAL, Portland State University — Numerical solutions have been found through a characteric equation. This equation was first obtained via similarity analysis of the equations of motion. The equations are analyzed for limiting pressure gradient cases, namely quasi-laminarization and separation. It is also found that the profiles for the laminar case and experimental data for a quasi-laminarized case do not collapse thus showing the effects and remnants of turbulence. The experimental data considered was first reported by Warnack and Fernholz as well as Jones et al. The equation contains two key parameters which are the pressure parameter as obtained via similarity analysis as well as the Pohlhausen parameter. Further investigation is done for the skin friction coefficient.

2:18PM R28.00007 Surface waves generated by a moving electromagnetic force on electrolytes[1] , GERARDO ALCALÁ, SERGIO CUEVAS, CIE UNAM, TERMOCIENCIAS TEAM — We present an experimental study of the gravity-capillary waves generated in the liquid-air interface of a shallow electrolytic layer ($NaHCO_3$) due to an electromagnetic force created by the interaction an applied direct electric current and a traveling magnetic field. The field is generated by a permanent magnet traveling in straight line externally to the bottom of the fluid container, and having a dominant component perpendicular to the plane spanned by the surface at equilibrium. Through long parallel electrodes, the current is applied transversally to the motion of the magnet so that the electromagnetic force points in favor or against the movement of the magnet, according to the polarity of either the electrodes or the magnet. The electromagnetic force acts as an obstacle for the flow (a magnetic obstacle) and, analogously to a moving solid body, generates a stationary wave pattern, which is reconstructed for various speeds of the magnet using optical methods. Differences in wave patterns are discussed for when the electromagnetic force points either in favor or against the movement of the magnet.

[1]Acknowledgment. Support form CONACYT through Project 131399 is thankfully acknowledged.

2:31PM R28.00008 Avoid Earth Extinction , DAYONG CAO, Avoid Earth Extinction Association — In 2011, the author supposes: the dark hole will take the dark comet to impact our solar system in 20 years. (see Dayong Cao, BAPS.2011.DFD.LA.24, BAPS.2012.APR.K1.78 and BAPS.2011.APR.K1.17) Asteroid 2011 AG5 will impact on Earth in 2040. (See Donald K. Yoemans, "Asteroid 2011 AG5 - A Reality Check," NASA-JPL, 2012) The dark Asteroid 2011 AG5 (as a dark comet) is made of the dark matte. Sun and its companion-dark hole are a binary system (Their systemic model-SDS for short there in after). The dark hole has a dark comet belt. The dark hole and dark comet are made of the dark matter which has a space-time (as frequence-amplitude square) center- a different systemic model from solar systemic model. Because it absorb the space-time and wave. So it is "dark." When the dark hole goes near the sun every 25-27 million years, it will take its dark comet belt to go into the solar system to impact our earth. In a other hand, it can change all of our systemic model and code which are controled by the SDS, such as the orbit both of the asteroid belt and planet (such as Jupiter), our atomic structure and our genetic code. It can trigger periodic Mass Extinction. We will use the dark matter to change the SDS to avoid forthcoming extinction.

Tuesday, November 20, 2012 1:00PM - 3:23PM –

Session R29 Chaos, Fractals, and Dynamical Systems III 32B - Chair: Nicholas Ouellette, Yale University

1:00PM R29.00001 Almost cyclic sets, topological chaos, and mixing in a lid-driven cavity flow , PRADEEP RAO, MOHSEN GHEISARIEHA, SHANE ROSS, MARK STREMLER, Virginia Tech — Topological chaos, or chaos that is guaranteed to exist in a system due to sufficiently complex motion of a few periodic orbits, has been demonstrated for a variety of flows, with a primary focus on creeping or ideal flows. Nearly-periodic systems can by analyzed in a similar way based on the presence of "braiding" Almost Cyclic Sets (ACS) with similarly complex space-time trajectories. For flow in a two-dimensional lid-driven cavity, this analysis can also be accurately extended to finite Reynolds numbers using a 2D Fourier-Chebyshev spectral algorithm for the streamfunction-vorticity formulation. We investigate the connection between the occurrence of braiding ACS, exponential stretching of material lines associated with topological chaos, and the efficiency of mixing for laminar flow in a lid-driven cavity.

1:13PM R29.00002 Acceleration feature points of unsteady shear flows[1] , JENS KASTEN, University of Leipzig, Germany, JAN REININGHAUS, IST, Austria, INGRID HOTZ, HANS-CHRISTIAN HEGE, ZIB, Germany, BERND R. NOACK, GUILLAUME DAVILLER, PIERRE COMTE, Institute PPRIME, France, MAREK MORZYŃSKI, Poznan University of Technology, Poland — We generalize velocity topology with centers (vortices) and saddle points in a Galilean-invariant manner. In particular, a computationally robust (derivative-free) framework for their extraction of two-dimensional unsteady flows is presented. The key enabler is the definition of feature points based on the acceleration magnitude. The extracted feature points are tracked over time resulting in corresponding trajectories. Using homological persistence and lifetime of features, a spatiotemporal importance measure for vortex core lines is introduced that enables a hierarchical filtering. As example, homological persistence is shown to discriminate between hydrodynamic and aeroacoustic flow structures. Our framework is applied to analytic examples as well as simulations of a cylinder wake, of a two-dimensional mixing layer and of a jet.

[1]Partially supported by the ANR Chair of Excellence TUCOROM and the German Research Foundation

1:26PM R29.00003 Perturbation of coherent structures in three-dimensional laminar flows: predictions versus experimental observations , FAN WU, Temple University, Philadelphia, MICHEL SPEETJENS, Eindhoven University of Technology, Eindhoven, DMITRI VAINCHTEIN, Temple University, Philadelphia, RUBEN TRIELING, HERMAN CLERCX, Eindhoven University of Technology, Eindhoven — Coherent structures in the fluid trajectories of three-dimensional (3D) laminar flows are key to their transport properties These structures typically undergo qualitative changes upon introducing some geometric or dynamical perturbation However, insight into such response scenarios in realistic 3D flows remains limited. The present study seeks to deepen this by investigating the response of coherent structures within a 3D time-periodic lid-driven cylinder flow in its Stokes limit to various weak perturbations. Numerical predictions by a spectral flow solver are compared against experiments by 3D PTV. The computations consider perturbation by weak fluid inertia and by a slight asymmetry in flow forcing, both causing essentially the same change in coherent structures. This signifies, consistent with theory on idealized flows, a generic response to weak perturbations, irrespective of their particular nature. The experiments, instead of explicit perturbation, rely on natural departures from a perfect state as e.g. geometric imperfections and weak fluid inertia. This results in dynamics that closely agree with the numerical predictions, thus offering first experimental evidence that indeed a universal mechanism is at play in the response of 3D coherent structures to perturbations

1:39PM R29.00004 Effect of the forcing on "steady" turbulent states , BRICE SAINT-MICHEL, GUILLAUME MANCEL, BÉRENGÈRE DUBRULLE, ÉRIC HERBERT, FRANÇOIS DAVIAUD, CEA Saclay - Laboratoire SPHYNX — Turbulent systems are intrinsically out of equilibrium, and thus have no reason to respect the symmetries of their forcing. It is yet generally accepted that symmetries are "statistically" restored in turbulence. Von Kármán swirling flows, though, might display continuous transitions or hysteretic behaviour depending on the type of forcing when the impeller speed is imposed. In the latter case, turbulent steady states are found to depend on the history of the system, three states being – at least marginally – stable for perfectly symmetric forcing. We have recently investigated the effect of the forcing on this system. When torque is imposed to the impellers, a whole new dynamics region is accessible inside the hysteresis loop; our system becomes multistable, continuously transiting between a small number of localised states. We characterize the structures displayed by such states and examine what governs the dynamics between them.

1:52PM R29.00005 Streamline and vorticity topology of eruption from a boundary layer induced by a 2D vortex patch , MORTEN ANDERSEN, MORTEN BRONS, Technical University of Denmark, Department of Mathematics, MARK THOMPSON, Monash University, Mechanical and Aerospace Engineering — We investigate the flow field generated by a vortex patch near a wall. Secondary vortices are created and boundary layer eruption may occur for increasing time or Reynolds number. The stream line topology and the vorticity topology are investigated and compared motivated by the work of Kudela & Malecha, Fluid Dyn. Res. 2009. Keeping track of vortices is a widely used procedure to explain "what is going on" in a fluid. However, different measures may be used for identifying a vortex. We will compare two of them under simplified conditions namely in the case of two dimensional incompressible flow with constant third component of the velocity vector. In the vorticity formulation a vortex is identified as an extremum of the vorticity. In the stream function formulation, if an elliptic fixed point exists then a vortex exists. The coordinate system is moving with constant speed equal to the generating vortex speed in inviscid flow. We find that vortex creation occur by saddle - node bifurcations in the streamlines, not by pinching off as suggested by Kudela & Malecha. Close to the creation of vortices, good agreement between the vorticity structure and the streamline topology is observed. At later stages, this may break down and streamline centers may disappear even though a vor

2:05PM R29.00006 Bifurcations in bifurcations: a dynamical analysis of an impacting T-junction flow[1] , KEVIN CHEN, CLARENCE ROWLEY, HOWARD STONE, DANIELE VIGOLO, Princeton University, STEFAN RADL, Graz University of Technology — Pipe bifurcations are a common flow configuration, for instance, in industrial systems and blood vessels. The impacting flow through a T-junction can cause corrosion, damage, and even aneurysms. To complement ongoing particle-laden flow physics research on this geometry, we perform a local bifurcation analysis of the steady-state Navier–Stokes solutions. We carry out numerical continuation on the Reynolds number, using a combination of linear extrapolation and the Newton-GMRES algorithm. A supercritical pitchfork (i.e., symmetry-breaking) bifurcation occurs at $Re \approx 410$, at which the pair of counter-rotating vortices in the outflow pipes becomes asymmetric. A supercritical Hopf bifurcation occurs at $Re \approx 540$, at which the asymmetric steady-state solution becomes unstable, and a stable periodic orbit grows out of this equilibrium.

[1]Supported by the DOD NDSEG fellowship and the NSF GRFP.

2:18PM R29.00007 Collective motion of interacting particles in spatially coherent flow , NIDHI KHURANA, NICHOLAS T. OUELLETTE, Yale University — Previous studies have shown that background flows can significantly modify the dynamics of independent active particles. In this work, we investigate how a spatially coherent turbulent-like flow modifies the collective behavior of interacting particles. We consider spherical point-like particles that interact according to a standard collective motion model. The particles move with a constant intrinsic speed but with a direction that depends on their neighbors. In addition, they are advected by a strongly fluctuating, multiscale flow field generated by kinematic simulation. By varying the relative strength of the intrinsic particle speed and the background flow, we study the effects of the complex flow field on the collective behavior of the particles.

2:31PM R29.00008 Efficient POD-based ROMs to approximate bifurcation diagrams , FILIPPO TERRAGNI, JOSE MANUEL VEGA, Universidad Politecnica de Madrid — Computing transitions and instabilities is a relevant issue in many fields, whose analysis usually involves time dependent nonlinear models. Thus, construction of bifurcation diagrams in extended systems may require huge computational resources. In dissipative problems, proper orthogonal decomposition (POD) may provide a low dimensional manifold containing the large-time dynamics of the system. In this talk, simple ideas are exploited in order to get flexible and accurate POD-based reduced order models (ROMs). The proposed method relies on the observation that POD manifolds resulting from snapshots calculated from a generic initial condition, a non-small time span, and specific values of the parameters contain the attractors for a wide range of parameter values. Appropriate POD manifolds can then be constructed with great flexibility and used to fast compute bifurcations. This is illustrated for fairly complex bifurcation diagrams (involving chaotic attractors) in the complex Ginzburg-Landau equation, in which a good set of snapshots can be calculated from either parameter values yielding simple dynamics, or rough numerical solvers, or different equations.

2:44PM R29.00009 Identifying Oscillatory Modes Using Harmonically Averaged Equations , JAY QI, JONATHAN TU, CLARENCE ROWLEY, Princeton University, RAJAT MITTAL, Johns Hopkins University — We present a method for analyzing dynamical systems exhibiting oscillatory behavior, using harmonic averaging. This method involves solving modified governing equations to directly obtain oscillatory modes corresponding to certain, specified frequencies. Common spectral analysis techniques post-process time-resolved data from full simulations; our approach instead leverages a priori knowledge of the system to directly compute the oscillatory modes. The method bears some similarity to a previous approach, the nonlinear frequency domain (NLFD) method, and is equivalent under certain conditions. However, because of the ability to choose arbitrary frequencies, harmonic averaging is advantageous in some cases, for instance for quasiperiodic phenomena, or when only a few frequencies are present. We demonstrate the method using a one-dimensional model problem, the Kuramoto-Sivashinsky equation, and show that the harmonic averaging method is able to accurately solve for the oscillatory modes in quasiperiodic systems.

2:57PM R29.00010 Developing flexible but efficient software for dynamical systems analysis of fluid flow[1] , SIAVASH AMELI, YOGIN DESAI[2], SHAWN SHADDEN, Illinois Institute of Technology — The computation of Lagrangian coherent structures (LCS) has become a standard tool for the analysis of advective transport in unsteady flow applications. LCS identification is typically accomplished by computation of finite-time (or finite-size) Lyapunov exponent fields (FTLE), or similar measures based on the Cauchy Green deformation tensor. Sampling of such fields over the fluid domain requires the advection of large numbers of tracers, which can be computationally intensive, but presents a large degree of data parallelism. There is compelling need for software that provides a flexible interface for LCS computation from fluid flow data, while leveraging advances in parallel architectures for data processing. We will describe work on these fronts. Specifically, we discuss the use of the Visualization Toolkit (VTK) libraries as a foundation for object-oriented, polymorphic LCS computation, and how this framework can facilitate integration into powerful flow visualization software such as Paraview. We also discuss the development of CUDA-c and OpenCL GPU kernels, and multicore CPU implementation, for efficient parallel computation of the flow map. We demonstrate results of these implementations on large-scale computations involving millions of tracers on large unstructured grids.

[1]This work was supported by the National Science Foundation, award number 1047963.
[2]SA and YD contributed equally to this work.

3:10PM R29.00011 Analysis of Fluid Flows via Spectral Properties of the Koopman Operator , IGOR MEZIĆ, University of California, Santa Barbara — We discuss theory and applications of Koopman modes in fluid mechanics. Koopman mode decomposition is based on the fact that normal modes of linear oscillations have its natural analogue - Koopman modes - in the context of nonlinear dynamics. To pursue this analogy, one must change the representation of the system from the state-space representation to the dynamics governed by the linear Koopman operator on an infinite-dimensional space of observables. The analysis is based on spectral properties of the Koopman operator. The point spectrum corresponds to isolated frequencies of oscillation present in the fluid flow, and also to growth rates of stable and unstable modes. The continuous part of the spectrum corresponds to chaotic motion on the attractor. A theoretical method of computation of the spectrum and the associated Koopman modes is given, in terms of the Generalized Laplace Analysis. A computational alternative is given by Arnoldi-type methods, leading to the so-called Dynamic Mode Decomposition (DMD). Koopman mode theory is shown to unify and provide a rigorous background for a number of different concepts that have been advanced in fluid mechanics, including Global Mode Analysis, triple decomposition and Dynamic Mode Decomposition.

Tuesday, November 20, 2012 1:00PM - 2:57PM –

Session R30 General Fluids IV 33A - Chair: Roberto Camassa, University of North Carolina

1:00PM R30.00001 Are the wake angles of a duck and a ship really the same? , MARC RABAUD, FREDERIC MOISY, Laboratoire FAST, Universite Paris-Sud — The wake of a disturbance moving at the water surface, like a ship or a duck, owes its shape to the dispersive property of surface gravity waves. According to Kelvin's theory, it is widely accepted, and sometimes observed, that the wake angle is independent of the disturbance velocity, and given by $\sin^{-1}(1/3) = 19.4$ degrees. However, field observations often show much smaller angles for fast ships, down to 5 - 10 degrees. The angle of these narrow wakes is actually found to decrease as the inverse of the disturbance velocity, similarly to the Mach cone of a supersonic disturbance in a non-dispersive medium. We propose here a simple model for this transition from a Kelvin regime (at low Froude number) to a Mach regime (at large Froude number) — where the Froude number is based on the disturbance length. This model is confirmed by numerical simulations, reproducing the variety of wake patterns observed for disturbances of various size and velocity.

1:13PM R30.00002 At the end of a moving string , JAMES HANNA, CHRISTIAN SANTANGELO, University of Massachusetts, Amherst — We address a basic problem in the dynamics of flexible bodies: the propagation of a shape along a string and its reflection at a free boundary. Although the string equations– inertia balancing stress in an inextensible curve– are quite old, the only exact solutions known for non-trivial geometries are traveling waves with spatially uniform stress. Suitable for closed "lariats," these solutions are incompatible with a free end, where the stress must vanish. It is impossible to drag an open, flexible, curved string along its tangents. This is reflected in the unwrapping motion of a string or chain as it is pulled around an object, and has strong implications for slender structures in passive locomotion, whether industrial cables or the ribbons of rhythmic gymnastics. We consider planar dynamics restricted to time-independent, but spatially varying, stress. We find a new exact solution at a distance $\propto t^{\frac{4}{3}}$ from the free end; continuation to the end requires introduction of a secular error into the positions and velocities and a singularity in acceleration $\propto t^{-\frac{2}{3}}$ at the end, which appears to have a physical basis. This work is an early step towards understanding the dynamics of a wide class of industrial and natural thin-object systems.

1:26PM R30.00003 Flow-induced oscillations of non-uniform pipes conveying fluid , GARY HAN CHANG, YAHYA MODARRES-SADEGHI, University of Massachusetts, Amherst — We study the influence of non-uniformity in the pipe cross-section and flow velocity on the oscillations of a pipe conveying fluid. A plain pipe conveying fluid loses its stability by buckling or flutter, depending on the pipe's boundary condition and the system parameters. A uniform plain cantilevered pipe loses its stability by a Hopf bifurcation, leading to either planar or non-planar flutter for flow velocities beyond the critical flow velocity. By attaching a spring or an extra mass at the tip of the pipe, secondary instabilities are observed leading to quasiperiodic and chaotic oscillations. In this study, Hamilton's principle is used to derive nonlinear equations of motion for a pipe with non-uniform system properties along its length. The pipe cross-section and material properties as well as its flow velocity can vary along the length. The resulting equations are then solved using the Galerkin technique. The model can be used to study a tapered pipe or a pipe with a sudden change in geometry and also the effect of local stiffening, narrowing, and pressure drop. The effect of continuous external damping can also be considered. It is shown that these non-uniformities can have significant impact on the observed instabilities.

1:39PM R30.00004 Unsteady flow near front and rear stagnation points , DMITRY KOLOMENSKIY, CERFACS, Toulouse, France, KEITH MOFFATT, DAMTP, University of Cambridge, UK, MARIE FARGE, LMD-IPSL-CNRS, ENS Paris, France, KAI SCHNEIDER, M2P2-CNRS and Aix-Marseille University, France — We consider unsteady flows near stagnation points on a cylindrical body immersed in a viscous incompressible fluid. This problem admits similarity solutions, assuming a hyperbolic time evolution of the free-stream velocity. These are exact solutions of the Navier–Stokes equations, having a boundary-layer character similar to that of classical steady forward stagnation-point flow. The velocity profiles are obtained by numerical integration of a non-linear ordinary differential equation. A wide range of possible behaviour is revealed, depending on whether the flow in the far field is accelerating or decelerating, and depending on the flow direction. For the forward-flow situation, the solution is unique for the accelerating case, but bifurcates for modest deceleration, while for sufficient rapid deceleration there exists a one-parameter family of solutions. For the rear-flow situation, a unique solution exists (remarkably!) for sufficiently strong acceleration, and a one-parameter family again exists for sufficient strong deceleration.

1:52PM R30.00005 General Multi-Species Dynamical Density Functional Theory , BENJAMIN GODDARD, ANDREAS NOLD, NIKOS SAVVA, Department of Chemical Engineering, Imperial College London, GRIGORIOS A. PAVLIOTIS, Department of Mathematics, Imperial College London, SERAFIM KALLIADASIS, Department of Chemical Engineering, Imperial College London — We extend our recent study on the dynamics of single-species colloidal fluids in the full position-momentum phase space to the dynamics of multi-species colloidal fluids. We include both inertia and the full hydrodynamic interactions, which strongly influence the non-equilibrium properties of the system. For many-particle systems, the number of degrees of freedom prohibit a direct solution of the underlying stochastic equations and a reduced model is necessary. Under minimal assumptions, we derive a dynamical density functional theory (DDFT), i.e. a reduction to the dynamics of the reduced one-body distribution. Via computations based on spectral methods extended to integral operators, we demonstrate the excellent agreement between this DDFT and the full Langevin equations for a range of multi-species systems. In suitable limits we recover existing DDFTs (which neglect inertia and/or hydrodynamic interactions) and we investigate the resulting corrections to these DDFTs.

2:05PM R30.00006 Controlling the Dynamics of the 2-D Navier-Stokes Equations , NEJIB SMAOUI, MOHAMED ZRIBI, Kuwait University — The dynamics of the two-dimensional (2-d) Navier-Stokes (N-S) equations with spatially periodic and temporally steady forcing $f = (\frac{1}{Re}k^3 \sin ky, 0)$ is analyzed. First, a system of nine-dimensional nonlinear dynamical system is obtained by a truncation of the 2-d N-S equations for various values of k. We show that for $k = 4$, the dynamics transforms from periodic solutions to chaotic attractors through a sequence of bifurcations including a period doubling scenarios. Then, a state feedback control is designed to drive the state of the system to any desired state.

2:18PM R30.00007 A New Approach to Model Order Reduction of the Navier-Stokes Equations[1] , MACIEJ BALAJEWICZ, EARL DOWELL, Duke University, BERND NOACK, Institute PPRIME — A new approach to model order reduction of the Navier-Stokes equations is proposed. Unlike traditional approaches, this method does not rely on empirical turbulence modeling or modification of the Navier-Stokes equations. It provides spatial basis functions different from the usual proper orthogonal decomposition basis function in that, in addition to optimally representing the solution, the new basis functions also provide stable reduced-order models. The proposed approach is illustrated with two test cases: two-dimensional flow inside a square lid-driven cavity and a two-dimensional mixing layer.

[1]Partially supported by the ANR Chair of Excellence TUCOROM

2:31PM R30.00008 Single series skewness representation for passive scalar advection in laminar pipe and channel flow[1] , ROBERTO CAMASSA, University of North Carolina at Chapel Hill, Mathematics, FRANCESCA BERNARDI, Politecnico di Milano, Nuclear Engineering, RICHARD MCLAUGHLIN, KEITH MERTENS, University of North Carolina at Chapel Hill, Mathematics — 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. Small and long time asymptotics will be discussed for the first three moments in both channel and pipe geometries in steady Poiseuille flows. Comparisons of theory with Monte Carlo simulations and preliminary experiments exhibit differences between channel and pipe flows in the skewness evolution.

[1]This research is supported by NSF RTG DMS-0502266, NSF RTG DMS-0943851, NSF RAPID CBET-1045653, NSF CMG ARC-1025523, NSF DMS-1009750, and ONR DURIP N00014-09-1-0840.

2:44PM R30.00009 Friction drag reduction by air cavities , OLEKSANDR ZVERKHOVSKYI, RENÉ DELFOS, TU Delft, TOM VAN TERWISGA, TU Delft/MARIN, JERRY WESTERWEEL, TU Delft — Air lubrication is investigated, as it is potentially one of the most efficient frictional drag reduction technologies for ships. Unlike bubbles in the boundary layer, which are explained to be effective while reducing the density of the liquid, artificial air cavities underneath a ship may reduce the amount of wetted surface, which presumable reduces more drag than the extra drag created by the devices required to create the cavities. The efficiency of such cavities has been studied experimentally on laboratory scale in a medium-speed water tunnel containing an optically accessible test section, equipped with a PIV system and cameras to obtain boundary layer- and cavity characteristics, and a force balance to measure the drag. The results of this study confirm that the drag reduction is proportional to the amount of non-wetted area. Based on experimental observations, a design criterion is presented for obtaining stable cavities with a low rate of air consumption. Furthermore, the experiments give insight into the formation and stability of air cavities, including the interaction between a series of cavities along the streamwise direction.

Tuesday, November 20, 2012 1:00PM - 2:57PM –

Session R31 Wind Energy III 33B - Chair: John Dabiri, California Institute of Technology

1:00PM R31.00001 Thermal stratification effects on a 4x3 wind turbine array boundary layer[1] , ELIZABETH CAMP, Portland State University, Department of Mechanical and Materials Engineering, MURAT TUTKUN, Norwegian Defense Research Establishment, RAUL BAYOAN CAL, Portland State University, Department of Mechanical and Materials Engineering — Efforts have intensified to investigate the effects of thermal stratification on wind turbine performance from a fluid mechanics the perspective. Recently, it has been shown that power production is highly dependent on the temperature variation of the atmospheric turbulent boundary layer. Hence, flow within a 4x3 turbine array is studied under neutral, unstable and stable thermal stratification regimes. The flow upstream of the model wind turbine array was modified using an active grid, strakes, roughness elements, and thermally controlled floor panels. In this wind tunnel experiment, the mean velocities, Reynolds stresses, and trends in C_p curves were measured along the centerline of the array using Particle Image Velocimetry in conjunction with torque sensors. The measurements consequently yield the flow development as well as the differences produced by the thermal effects within the array.

[1]This work is funded by the National Science Foundation grants CBET-1034581 and ECCS-1032647 as well as Institute for Sustainable Solutions at Portland State University.

1:13PM R31.00002 Statistical analysis of kinetic energy entrainment in a model wind turbine array boundary layer[1] , RAUL BAYOAN CAL, NICHOLAS HAMILTON, Portland State University, HYUNG-SUK KANG, US Naval Academy, CHARLES MENEVEAU, Johns Hopkins University — For large wind farms, kinetic energy must be entrained from the flow above the wind turbines to replenish wakes and enable power extraction in the array. Various statistical features of turbulence causing vertical entrainment of mean-flow kinetic energy are studied using hot-wire velocimetry data taken in a model wind farm in a scaled wind tunnel experiment. Conditional statistics and spectral decompositions are employed to characterize the most relevant turbulent flow structures and determine their length-scales. Sweep and ejection events are shown to be the largest contributors to the vertical kinetic energy flux, although their relative contribution depends upon the location in the wake. Sweeps are shown to be dominant in the region above the wind turbine array. A spectral analysis of the data shows that large scales of the flow, about the size of the rotor diameter in length or larger, dominate the vertical entrainment. The flow is more incoherent below the array, causing decreased vertical fluxes there. The results show that improving the rate of vertical kinetic energy entrainment into wind turbine arrays is a standing challenge and would require modifying the large-scale structures of the flow.

[1]This work was funded in part by the National Science Foundation (CBET-0730922, CBET-1133800 and CBET-0953053).

1:26PM R31.00003 Effects of downstream and cross-stream spacings in an array of aligned wind turbines , DANIEL HOUCK, MATT MELIUS, RAUL CAL, Portland State University — This study seeks to address the effects of downstream and cross-stream spacings in a finite array of aligned wind turbines. A series of wind tunnel experiments were performed in which a 3x4 array of scale wind turbines were arranged in four patterns of different downstream (DS) and cross-stream (CS) spacings. The four cases, using the rotor diameter as unity, were: 6DSx3CS, 6DSx1.5CS, 3DSx3CS, and 3DSx1.5CS. All cases were subjected to the same inflow conditions, which replicated an atmospheric boundary layer. Particle image velocimetry was used to measure the flow field at the entrances and exits of the first and third rows of the array and velocity field data was collected. Results indicate effects of the downstream and cross-stream spacings separately, but also effects that appear to correlate to turbine density and not a specific arrangement. The cases often pair up with similar trends, but do so differently in the first and third rows and even, at times, in the entrances and exits and at different heights. The effects of cross-stream spacing are more pronounced in the first row than the third, and vice versa for downstream spacing.

1:39PM R31.00004 Flow development comparison in two-bladed and three-bladed model wind turbine arrays , C. DALTON MCKEON, JONATHAN SULLIVAN, Texas Tech University, ELIZABETH CAMP, MATTHEW MELIUS, DOMINIC DELUCIA, RAUL BAYOAN CAL, Portland State University, LUCIANO CASTILLO, Texas Tech University and the National Wind Resource Center — Vertical entrainment of energy through turbulent structures is compared between two-bladed and three-bladed model wind turbine arrays. A wind tunnel study under neutrally stratified conditions has been performed to compare the differences in large-scale structures of energy fluxes in two 3×4 arrays. Both arrays have three turbines with 3D spacing in the spanwise direction and four turbines with 6D spacing in the streamwise direction. The rotor diameter for both is 12 cm. The same mean velocity at hub height is maintained for both arrays. The power coefficient for both models is matched, resulting in different tip speed ratios. Consequently, both arrays of turbines are extracting energy from the flow at the same rate, which results in the identification of differences in energy fluxes due to the distinct number of blades on the rotor. Velocity data is collected via stereoscopic PIV; planes are located along the centerline of the array and are parallel with the streamwise direction. Profiles of mean velocity, Reynolds stresses, energy flux, and energy dissipation are generated. These profiles are used to compare the mechanisms of energy exchange in the two-bladed and three-bladed arrays.

1:52PM R31.00005 Markovian properties of wind turbine wakes within a 3x3 array , MATTHEW MELIUS, Portland State University, MURAT TUTKUN, Norwegian Defense Research Establishment, RAÚL BAYOÁN CAL, Portland State University — Wind turbine arrays have proven to be significant sources of renewable energy. Accurate projections of energy production is difficult to achieve because the wake of a wind turbine is highly intermittent and turbulent. Seeking to further the understanding of the downstream propagation of wind turbine wakes, a stochastic analysis of experimentally obtained turbulent flow data behind a wind turbine was performed. A 3x3 wind turbine array was constructed in the test section of a recirculating wind tunnel where X-wire anemometers were used to collect point velocity statistics. In this work, mathematics of the theory of Markovian processes are applied to obtain a statistical description of longitudinal velocity increments inside the turbine wake using conditional probability density functions. Our results indicate an existence of Markovian properties at scales on the order of the Taylor microscale, λ, which has also been observed and documented in different turbulent flows. This leads to characterization of the multi-point description of the wind turbine wakes using the most recent states of the flow.

2:05PM R31.00006 Low-order flow modeling of vertical-axis wind turbine arrays[1] , DANIEL ARAYA, California Institute of Technology, ANNA CRAIG, Stanford University, JOHN DABIRI, California Institute of Technology — We present a potential flow model of recent experimental measurements of a full-scale array of vertical-axis wind turbines. Potential flow elements are used to approximate the flow physics of the array. Average velocity measurements, taken over the course of several months from different locations within the array, are used to compute velocity residuals, which are minimized to find the best-fit model. In addition, we present an approach to extend the empirical model to larger turbine array sizes by deducing the relation of the local flow velocity to the potential flow element strengths. This low-order modeling approach has the advantage of being simple enough for rapid optimization of small turbine arrays, yet robust enough for also at least qualitatively predicting the performance of larger arrays.

[1]This work was partially supported by the Department of Defense (DoD) through the National Defense Science & Engineering Graduate Fellowship (NDSEG) Program.

2:18PM R31.00007 Low dimensional model of energy reconstruction for inline and offset wind turbine arrays , NICHOLAS HAMILTON, Portland State University, MURAT TUTKUN, Norwegian Defence Research Establishment (FFI), RAUL BAYOAN CAL, Portland State University — Wind tunnel data was obtained via stereo-Particle Image Velocimetry for wind turbine models in rectangular and row-offset array configurations. Inflow and wakes were compared statistically and through proper orthogonal decomposition (POD). Spacing within the array configurations lead to varied wake recovery conditions visible in statistics and POD analyses. The number of snapshot POD modes required to reconstruct specified energy levels for the wakes demonstrates a strong dependence on the array configuration. Quantities including flux of kinetic energy and production rebuild with fewer POD modes than vorticity and dissipation as they rely on large-scale structures for shape andintensity. Energy content and organization in small-scale structures contribute to the delay of recovery of the flow to upstream inflow conditions.

2:31PM R31.00008 Experimental study of flow around scaled wind turbine arrays[1] , RAMIRO CHAVEZ ALARCON, B.J. BALAKUMAR, FANGJUN SHU, New Mexico State University — An array of scaled model wind turbines, designed using blade element momentum theory, were investigated in a low-speed wind tunnel under uniform laminar inflow conditions. To investigate the influence of neighbor wind turbines, a 2D particle image velocimetry (PIV) system was used to measure the flow field in the wake of the turbines. It is found that flow in the wake of the turbine is axisymmetric about the turbine axis, and is not dependent on the blades orientation. A detailed dataset containing wake structure variations including velocity deficit and Reynolds stresses was obtained and compared with previously obtained data corresponding to a single wind turbine. The power extracted by a single wind turbine under the influence of the array was obtained from the velocity deficit observed at the wake and compared with the power obtained from torque sensor measurements. The complete investigation is compared with previously obtained data corresponding to a single wind turbine.

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

2:44PM R31.00009 Performance and wake measurements for a cross-flow axis lift-driven turbine , PETER BACHANT, MARTIN WOSNIK, University of New Hampshire — A 2.67 m wide, 3.67 m deep tow tank was used to measure performance and wake flow from a cross-flow axis (CFA) lift-driven turbine. A custom turbine test bed was designed that allows precise control of turbine tip speed ratio via servomotor, as well as measurements of power and overall drag on the turbine. Blade forces are measured with high spatio-temporal resolution via piezoelectric film sensors. The flow-field in the near wake is measured via high frame rate PIV and acoustic Doppler velocimetry. The size of the turbine model and a newly renovated tow tank mechanism allowed the blade chord Reynolds number to be roughly an order of magnitude higher than in previous studies. The overall goal of this study is to accurately measure fluid-blade interactions to observe effects of design parameter changes to performance and wake structure; also to help validate (or invalidate) numerical models. The higher Reynolds numbers of these results, especially those under dynamic stall conditions, make them more applicable to full scale commercial installations.

Tuesday, November 20, 2012 1:00PM - 2:57PM –

Session R32 Granular Flows V 33C - Chair: Troy Shinbrot, Rutgers University

1:00PM R32.00001 A numerical study of unsteady shear flows of fluid-saturated granular materials in the presence of gravity , CHRISTOS VARSAKELIS, MILTIADIS PAPALEXANDRIS, Université catholique de Louvain — In this talk we present results from a numerical study of unsteady, shear flows of fluid-saturated granular materials in the presence of gravity. In our study, we employ a two-pressure, two-velocity continuum model for the mixtures of interest. The governing equations are integrated via a predictor-corrector algorithm that combines a projection method for the pressure of each phase and an interface-tracking scheme. Initially, a high particle concentration ball is placed between two parallel plates while the rest of the domain is filled with a carrier fluid. The mixture is set in motion by the horizontal movement of the upper plate with constant speed. Because of the developing shear stresses and the onset of the Rayleigh-Taylor instability, the ball deforms to a wavy finger-like shape whose length increases with time. Further, fluid entrainment produces a mushroom pattern in its frontal part. At the same time, this granular finger descends due to gravity and once it reaches the bottom plate it forms an asymmetric granular pile. This talk concludes with results from a parametric study with respect to the shear rate and the diameter of the particles.

1:13PM R32.00002 ABSTRACT MOVED TO H32.00004 –

1:26PM R32.00003 Erosion dynamics of a wet granular medium , GAUTIER LEFEBVRE, PIERRE JOP, Surface du Verre et Interfaces UMR 125 CNRS/Saint-Gobain, HETEROGENEOUS REACTIVE MATERIALS TEAM — Liquid may give strong cohesion properties to the granular medium, and confers a solid-like behavior. We study the evolution of a fixed aggregate of wet granular matter subjected to a flow of dry grains. In the confined geometry of a thin cell, the aggregate is hold by the walls, and the dry matter flowing around will pull grains out of the aggregate. Thus, by granular erosion, the interface is modified. Image treatment allows us to follow the shape of the aggregate, and to quantify the erosion speed. We have set-up two configurations of erosion. At the center of a half-filled rotating drum, we introduce the wet material with a determined liquid content. During the rotation, the dry grains flow around the fixed obstacle and grains are pulled-out of the aggregate, reducing its size. This provides an erosion rate, related to the liquid and grains physical properties. We analyze the influence of liquid properties (surface tension, viscosity) and quantity in this geometry. A model based on the fluctuations of the flow explains the dependencies observed in the experiment. In an open cell between vertical plates, we can form a heap-shaped aggregate. Then, with a funnel of constant outlet, we inject dry grains, flowing on top of the cohesive heap. We observe destabilization of an initial flat profile in certain conditions. The coupling between the flow stress and the shape of the heap creates periodical structures, which propagate to the top through the erosion process.

1:39PM R32.00004 An introduction to the Hele-Shaw beach experiments , ANTHONY THORNTON, BRAM VAN DER HORN, DEVARAJ VAN DER MEER, University of Twente, WOUT ZWEERS, FabLab, ONNO BOKHOVE, University of Twente — The sea, as well as being a destructive force can also be constructive and can move great quantities of sand often forming a beach. Waves can move material both up and down the beach, leading to the construction of sloping beaches. Wave-sand dynamics are studied via experiments. The tank is narrow, just over one-particle diameter wide, creating a quasi-2D set-up also geared towards mathematical modelling. There is strong two-way feedback between the free-surface waves and the beach morphology. The waves transport the particles, changing the basal topography, causing the waves to transform from rolling to breaking. "All" classical breaker types (plunging, collapsing, spilling and surging) are observed on a time-scale of about a second. Finally, on longer time-scales many steady beach morphologies are observed, including dry and wet beaches, dry berms/dunes, and bars. The highlight being dry dunes which have dynamic waves crashing on the seaward-side and quiescent water on the far side.

1:52PM R32.00005 Multiphase flow description of material pulverization[1] , DUAN ZHANG, XIA MA, BALAJI JAYARAMAN, Los Alamos National Laboratory — Material failure and crack growth are traditionally studied in solid mechanics. However, rapid material failure often results in growth of numerous cracks and pulverization of a material. Before pulverization, the motion of the material is described by the set of equations for solid. After pulverization, if the size of the debris piece is sufficiently small, the effect of surrounding media, such as air and water, is important, and the motion of the material is often modeled as a disperse multiphase flow. Numerical simulation of the process encounters two significant challenges. The first challenge is quite practical. That is how to interface a solid code with a fluid code. The second challenge is more subtle and difficult. That is how to describe the transition from a continuum to a granular state, and provide a proper initial condition for the multiphase calculation. In this talk, we first introduce a framework of equations capable of describing the entire pulverization process based on multiphase flow formulation, and then a numerical method capable of unifying solid and fluid calculations. Despite the need to study suitable closure models for the equations, in this talk we present a few numerical examples that have be obtained by using simple closure relations.

[1]This work was performed under the auspices of the United States Department of Energy.

2:05PM R32.00006 Fast X-ray Imaging Applications for Granular Physics , YUJIE WANG, YIXIN CAO, CHENGJIE XIA, BINQUAN KOU, HAOHUA SUN, 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 60439, USA — Studying granular systems with x-ray imaging technique, including x-ray computed tomography (CT) and ultrafast x-ray projection imaging, has great superiority. Due to the penetrating properties of x-ray, internal structures of a granular system could be obtained. Using x-ray CT technology, we studied packing problems with various granular systems, such as mono-dispersed hard spheres, wet spheres, rods, poly-dispersed foams, etc. At the same time, ultrafast x-ray phase contrast imaging technique provides a projective realization of evolving systems, which is one of the few experimental methods that can probe dynamic properties of granular systems. These experimental works will contribute to revealing some important properties of granular systems.

2:18PM R32.00007 Two-level hierarchical structure in nano-powder agglomerates in gas media

, LILIAN DE MARTIN, WIM G. BOUWMAN, J. RUUD VAN OMMEN, Delft University of Technology — Nanoparticles in high concentration in a gas form agglomerates due to the interparticle van der Waals forces. The size and the internal structure of these nanoparticles agglomerates strongly influence their dynamics and their interaction with other objects. This information is crucial, for example, when studying inhalation of nanoparticles. It is common to model the structure of these agglomerates using a fractal approach and to compare their dimension with the dimension obtained from aggregation models, such diffusion limited aggregation (DLA). In this work we have analyzed the structure of nanoparticles agglomerates in situ by means of Spin-Echo Small-Angle Neutron Scattering (SESANS), while they were fluidized in a gas stream. The advantage of SESANS over conventional SANS is that SESANS can measure scales up to 20 microns, while SANS does not exceed a few hundred of nanometers. We have observed that when agglomerates interact, their structure cannot be characterized by using only one scaling parameter, the fractal dimension. We have found that there are at least two structure levels in the agglomerates and hence, we need at least two parameters to describe the autocorrelation function in each level.

2:31PM R32.00008 Shock Wave Instability in Dissipative Granular Gases

, NICK SIRMAS, MATEI RADULESCU, University of Ottawa — The current study addresses the stability of shock waves propagating through dissipative granular gases. We perform molecular dynamics simulations of colliding hard disks accelerated by a piston. The collisions between the particles are modeled with a constant coefficient of restitution for activated collisions. An activated state is first established through shock compression, followed by a relaxation period until equilibrium is reached. Due to the existence of the activation threshold, the compacted region retains some thermal motion. Our numerical experiments reveal that the structure of these shock waves is unstable. Distinctive high density non-uniformities are formed, which take the form of convective rolls. We find that the characteristic spacing between the bumps is correlated with the relaxation length scale, which is dependent on the coefficient of restitution and shock strength. The results are also investigated in the framework of shock wave theory. Using analytical and numerical results for the shock Hugoniot, we show that both D'yakov-Kontorovich instability, and Bethe-Zel'dovich-Thompson instability can be ruled out. Instead, the results suggest that the clustering instability of Goldhirsch and Zanetti is the dominant mechanism controlling the shock instability.

2:44PM R32.00009 Clustering Instabilities in Homogeneously Cooling Particulate Flows

, PETER MITRANO, STEVEN DAHL, JOHN ZENK, CHRISTOPHER EWASKO, CHRISTINE HRENYA, University of Colorado at Boulder — Particulate flow instabilities, such as particle clustering, are commonly observed in industrial applications (e.g., gasifiers and fluidized beds). The particle dynamics associated with such instabilities have been studied through experiment, theory, and discrete-particle simulation. However, most previous theoretical analysis has been limited to linear stability analyses, and no quantitative predictions about instabilities have been obtained via numerical simulations of hydrodynamic models or via direct simulation Monte Carlo. In this work, we use a combination of numerical hydrodynamic simulations, linear stability analyses, and discrete-particle simulations to quantitatively assess the ability of hydrodynamics to describe instabilities in particulate flows. We find excellent agreement between discrete-particle simulations and hydrodynamic simulations for the onset of particle clustering. Such agreement demonstrates the aptitude of the Enskog equation in describing particulate flows and (since velocity gradients exist) the versatility of the small-Knudsen-number expansion. A systematic under prediction of clustering onset by linear analyses exemplifies the importance of nonlinear mechanisms (e.g., viscous heating) in cluster formation.