St. Anthony Falls Laboratory Research

at the

AMERICAN PHYSICAL SOCIETY

66TH ANNUAL DFD MEETING

Sunday, November 24 – Tuesday, November 26, 2013

Pittsburgh, Pennsylvania

St. Anthony Falls Laboratory. The St. Anthony Falls Laboratory (SAFL) is an interdisciplinary fluid mechanics research and educational facility of the College of Science and Engineering at the University of Minnesota. Located on Hennepin Island in the Mississippi River in the heart of Minneapolis, SAFL research explores science-based solutions to major societal challenges in energy, the environment and health. Learn more at www.safl.umn.edu.

UNIVERSITY OF MINNESOTA
Driven to Discover℠

ORAL SESSIONS

SUNDAY, NOVEMBER 24, 2013

Sunday, November 24, 2013
Session A13: Focus Session: Marine Hydrokinetic Energy Conversion I
8:00 AM–8:13 AM
Room: 301

Interaction between an axial-flow model hydrokinetic turbine and an erodible channel (Abstract: A13.00001)

Craig Hill (University of Minnesota); Mirko Musa (University of Trento); Leonardo P. Chamorro (University of Illinois); Michele Guala (University of Minnesota)

Laboratory experiments were carried out to examine the effect of relatively large-scale bedforms on the performance of a model axial-flow hydrokinetic turbine. The turbine rotor, *dT*=0.15m, was attached to a miniature DC motor, and allowed for voltage data acquisition at 200 Hz along with 3D hub-height inflow velocity, *Uhub*, approximately 7*dT* upstream of the turbine. Spatio-temporal bed elevations were acquired along three longitudinal sections and at least one transverse transect within the flume providing the temporally-averaged scour and deposition patterns characterizing the turbine near-field region. Turbine-turbine interaction was investigated under aligned configurations in the streamwise direction with variable spacing both in clear water scour and live bed transport conditions. Effects from both migrating bedforms and the upstream turbine were observed in the long-term and short-term voltage fluctuations of the downstream turbine. Combined measurements of inflow velocity, bed topography and turbine voltage were used to obtain joint statistics and correlations, which provided an indication of the variability in environmental exposure and performance that hydrokinetic turbines will encounter in natural erodible rivers.

Sunday, November 24, 2013
Session A16: Biofluids: Physiological I - Computational Studies in Cardiovascular Flows
8:52 AM–9:05 AM
Room: 304

Patient-specific simulation of a trileaflet aortic heart valve in a realistic left ventricle and aorta (Abstract: A16.00005)

Anvar Gilmanov (University of Minnesota); Trung Le (University of Minnesota); Henryk Stolarski (University of Minnesota); Fotis Sotiropoulos (University of Minnesota)

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

Sunday, November 24, 2013
Session A14: Experimental Techniques I: PIV Algorithms
9:18 AM–9:31 AM
Room: 302

Quantifying large-scale flow structures in the wake of a 2.5 MW wind turbine using natural snowfall (Abstract: A14.00007)

Jiarong Hong (University of Minnesota); Mostafa Toloui (University of Minnesota); Sean Riley (University of Minnesota); Michele Guala (University of Minnesota); Kevin Howard (University of Minnesota); Leonardo Chamorro (University of Illinois)
James Tucker (University of Minnesota); Fotis Sotiropoulos (University of Minnesota)

The atmospheric inflow conditions around utility-scale turbines and multi-turbine arrayed wind farms remain poorly known, despite ongoing research, resulting in considerable wind plant power loss and increased annual operating costs. Gaining detailed full-scale flow information is constrained by low resolution spatial characterization of the flow field around turbines due to a lack of utility-scale research facilities and a number of technical challenges associated with obtaining measurements. Taking advantage of natural snowfall, we now achieve velocity field measurements in the wake of a 2.5 MW wind turbine at a scale of 36x36 m2. The spatial and temporal resolutions of the measurements are sufficiently high to quantify the evolution of blade-generated coherent motions, such as the tip and trailing sheet vortices, identify their instability mechanisms, and correlate them with turbine operations, control, and performance. This technique has been further validated by comparing the obtained mean velocity and Reynolds stress profiles, up to 60 m above the ground with sonic anemometer measurements at specific elevations, where less than a 3{\%} and 10{\%} difference were observed, respectively.

ORAL SESSIONS

SUNDAY, NOVEMBER 24, 2013

Sunday, November 24, 2013 Session A11: Bubbles I: Cavitation, Nucleation and Ventilation 9:31 AM–9:44 AM Room: 335	**Ventilation of an hydrofoil wake** (Abstract: A11.00008) Roger Arndt (University of Minnesota); Seung Jae Lee (University of Minnesota); Garrett Monson (University of Minnesota); Ashish Karn (University of Minnesota); Jiarong Hong (University of Minnesota) Ventilation physics plays a role in a variety of important engineering applications. For example, hydroturbine ventilation is used for control of vibration and cavitation erosion and more recently for improving the dissolved oxygen content of the flow through the turbine. The latter technology has been the focus of an ongoing study involving the ventilation of an hydrofoil wake to determine the velocity and size distribution of bubbles in a bubbly wake. This was carried out by utilizing particle shadow velocimetry (PSV). This technique is a non-scattering approach that relies on direct in-line volume illumination by a pulsed source such as a light-emitting diode (LED). The data are compared with previous studies of ventilated flow. The theoretical results of Hinze suggest that a scaling relationship is possible that can lead to developing appropriate design parameters for a ventilation system.
Sunday, November 24, 2013 Session A11: Bubbles I: Cavitation, Nucleation and Ventilation 9:44 AM–9:57 AM Room: 335	**Improved Performance With Ventilation** (Abstract: A11.00009) Ellison Kawakami (3M Corporation); Seung Jae Lee (University of Minnesota); Ashish Karn (University of Minnesota); Jiarong Hong (University of Minnesota); Roger Arndt (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, ventilation is required to supply an artificial cavity until conditions at which a natural supercavity can be sustained are reached. Various aspects of the flow physics of a supercavitating vehicle have been under investigation for several years at Saint Anthony Falls Laboratory. Both steady flow and simulated flow below a wave train have been studied. Using a high speed camera and the proper software, it is possible to synchronize cavity dimensions with pressure measurements taken inside the cavity to permit an in-depth study of unsteadiness. It was found that flow unsteadiness caused a decrease in the overall length of the supercavity while having only a minimal effect on the maximum diameter. Results regarding supercavity shape, ventilation demand, cavitation parameters and closure methods are reviewed in light of new studies that focused on various closure mechanisms.
Sunday, November 24, 2013 Session D1: Geophysical: Atmospheric II 2:15 PM–2:28 PM Room: 323	**Influence of inflow condition on wind turbine operation and wake unsteadiness** (Abstract: D1.00001) Kevin Howard (University of Minnesota); Leonardo P. Chamorro (University of Illinois);Michele Guala (University of Minnesota) A model wind turbine was tested in a closed-circuit wind tunnel under three different inflow conditions, (i) smooth wall turbulent boundary layer, (ii) preceding turbine wake and (iii) three dimensional sinusoidal hill wake, and three thermal stratifications. Two particle image velocimetry (PIV) fields were taken simultaneously upwind and downwind of the turbine along with the turbine voltage, which quantifies rotor fluctuations. Both wall-normal PIV fields were oriented on the centerline of the turbine and captured flow data in a window of approximately 1.1D by 1.1D, where D is the rotor diameter of the turbine. The upwind PIV measured the changing inflow conditions while both the voltage and downwind PIV field provided data that describes the response of the turbine and near wake to the inflow, respectively. Changes occurring in the inflow, whether upwind perturbation or thermal stability related, were statistically linked to the turbine voltage production and wake unsteadiness, as shown by turbulence intensity and swirling strength contours. A laboratory to field scale comparison is completed by inspecting light detection and ranging (Lidar) data taken upwind of the EOLOS utility scale, 2.5 MW wind turbine in conjunction with the turbine power production time signals.

ORAL SESSIONS

SUNDAY, NOVEMBER 24, 2013

Sunday, November 24, 2013
Session E13: Focus Session: Marine Hydrokinetic Energy Conversion III
4:58 PM–5:11 PM
Room: 301

On the effects of turbine geometry on the far wake dynamics of an axial flow hydrokinetic turbine (Abstract: E13.00002)

Fotis Sotiropoulos (University of Minnesota); Xiaolei Yang (University of Minnesota); Seokkoo Kang (Hanyang University)

In large-eddy simulation (LES) of multi-turbine arrays actuator disk (AD) or actuator line (AL) models are employed to simulate individual turbines. Such parameterizations do not take into account the details of the turbine geometry and, therefore, cannot be expected to accurately resolve the flow in the near wake. We investigate the performance of AD and AL models by comparing their predictions with laboratory measurements and with LES resolving the geometrical details of the turbine. We simulate the flow past an axial flow hydrokinetic turbine in a fully-developed turbulent flow in an open channel using: turbine-geometry resolving LES (LES-TG) and LES-AD and LES-AL parameterizations. We show that LES-TG reveals very complex large-scale dynamics in the near wake, driven by the interaction of a counter-rotating to the turbine hub vortex and the top-tip shear layer, which appears to influence both the mean flow characteristics and the intensity of wake meandering several rotor diameters downstream. The LES-AD and LES-AL results cannot capture the geometry-induced complex near wake phenomena and yield flows that exhibit important differences with the LES-TG results in the far wake. The mechanisms that give rise to and modeling implications of these differences will be discussed.

Sunday, November 24, 2013
Session E13: Focus Session: Marine Hydrokinetic Energy Conversion III
5:11 PM–5:24 PM
Room: 301

Large-eddy simulation of the flow over a hydrokinetic turbine mounted on an erodible bed (Abstract: E13.00003)

Xiaolei Yang (University of Minnesota); Ali Khosronejad (University of Minnesota); Fotis Sotiropoulos (University of Minnesota)

Marine and hydrokinetic (MHK) energy comprises an important source of clean and renewable energy. The beds of natural waterways are usually erodible. The hydrokinetic turbines affect the sediment transport, which, on the other hand, also influences the performance of hydrokinetic turbines. A powerful computational framework for simulating marine and hydrokinetic (MHK) turbine arrays mounted in complex river bathymetry with sediment transport has been developed and validated by our group. In this work we apply this method to simulate the turbulent flow over a hydrokinetic turbine mounted in an open channel with erodible bed. Preliminary results show qualitatively good agreement with the experiment. Detailed comparison with measurements and analysis of the simulation results will be presented in the conference.

Sunday, November 24, 2013
Session E25: Flow Control III - Drag Reduction
5:24 PM–5:37 PM
Room: 320

Drag-Reduction Effectiveness of Riblet Films in Adverse Pressure Gradients (Abstract: E25.00004)

Aaron Boomsma (University of Minnesota); Fotis Sotiropoulos (University of Minnesota)

Riblet films are micro-grooved structures that are widely known to passively reduce skin friction. Past studies have almost solely focused on riblet performance in channel-flows. However, possible applications of riblets include wind turbine blades, gas turbine blades, and other complex bodies that are exposed to non-zero pressure gradient flows---specifically adverse pressure gradients. We use high-resolution large eddy simulations of turbulent flow over three-dimensional riblets under an adverse pressure gradient. We analyze the computed results to quantify drag reduction effectiveness for different riblet shapes and to examine pertinent turbulent structures to gain a fundamental understanding of riblet performance.

MONDAY, NOVEMBER 25, 2013

Monday, November 25, 2013
Session G28: Waves II
8:00 AM–8:13 AM
Room: Spirit of Pittsburgh Ballroom B/C

Scaling and kinematics of a floating wind turbine under ocean waves and variable thrust: an experimental study (Abstract: G28.00001)

Chris Feist (University of Minnesota); Kelley Ruehl (Sandia National Laboratories); Michele Guala (University of Minnesota)

Scale model wave channel experiments were performed to study the motion of an offshore floating wind turbine in operational sea states. The model tests were conducted on a 1:100 Froude scaled Sandia National Labs 13.2 MW prototype offshore wind turbine with a barge style floating platform. The platform is modeled after the MIT/NREL Shallow Drafted Barge designed for the 5MW Offshore Baseline wind turbine. The wave environment used in the model tests is representative of the deep-water sea states off the coast of Maine as well as the Pacific Northwest. The purpose of the tests is to validate a computational model of the turbine-wave interaction where the effects of airflow are not considered. To simplify the tests and validation, the platform motion is restricted to two degrees of freedom, pitch and heave, by attaching two roller support types at the center of gravity along the pitch axis. The major aerodynamic force acting on the turbine, i.e. the rotor thrust, is provided by spinning a scaled rotor at a controlled rotational speed. A subset of experiments were performed to study the effect of a mean or fluctuating rotor thrust on the platform dynamics, exploring strategies to control the thrust as a function of platform pitch angle and minimize platform oscillations.

ORAL SESSIONS

MONDAY, NOVEMBER 25, 2013

Monday, November 25, 2013
Session G28: Waves II
8:13 AM–8:26 AM
Room: Spirit of Pittsburgh Ballroom B/C

Fluid-structure interaction simulation of floating wind turbines interacting with complex, large-scale ocean waves (Abstract: G28.00002)

Antoni Calderer (University of Minnesota); Xin Guo (University of Minnesota); Lian Shen (University of Minnesota); Fotis Sotiropoulos (University of Minnesota)

We develop a numerical method for simulating coupled interactions of complex floating structures with large-scale ocean waves and atmospheric turbulence. The Fluid-Structure Interaction (FSI) solver integrates the curvilinear immersed boundary method of Borazjani et al. (JCP 2008) with the level-set method of Kang et al. (Adv. in Water Res. 2012) and is capable of simulating the coupled dynamic interaction of arbitrarily complex bodies with airflow and waves. The large-scale wave model is based on the two-fluid coupled approach of Yang et al. (JCP 2011), which employs a high-order spectral method for simulating the water motion and a viscous solver with undulatory boundaries for the air motion. The large-scale wave field solver is coupled with the near-field FSI solver by feeding into the latter large-scale waves via the pressure-forcing method of Guo et al. (JCP 2009), appropriately adapted herein for the level set method. We validate the model under both simple wave trains and three-dimensional directional waves and compare the results with experimental and theoretical solutions. Finally, we demonstrate the capabilities of the new solver by carrying out large eddy simulation of a floating offshore wind turbine platform interacting with realistic ocean waves.

Monday, November 25, 2013
Session G24: Aerodynamics II
9:31 AM–9:44 AM
Room: 319

Numerical study of ocean wave effect on offshore wind farm (Abstract: G24.00008)

Lian Shen (University of Minnesota); Di Yang (Johns Hopkins University); Charles Meneveau (Johns Hopkins University)

Wind power at sea has become increasingly important in renewable energy study. For energy harvesting, winds over oceans have many advantages over winds on land, for example, larger and open surface area, faster wind speed, and more wind resource close to high population regions. On the other hand, the presence of ocean waves introduces complexities to wind turbines. There is a critical need to study the dynamical interactions among marine atmospheric boundary layer, ocean wave field, and floating turbines. In this research, we study offshore wind farm by performing large-eddy simulations for winds coupled with potential-flow-theory based simulations for broadband irregular waves, with the wind turbines represented by an actuator disk model. Our results show that windseas at different development stages result in different sea-surface roughness and have an appreciable effect on wind profile and the energy extraction rate of the turbines. If swells are present, swell-to-wind momentum and energy transfer further changes the wind field to introduce oscillations in as well as modify the mean of the wind power.

Monday, November 25, 2013
Session G5: CFD IV
9:44 AM–9:57 AM
Room: 327

Parallel Cartesian grid refinement for 3D complex flow simulations (Abstract: G5.00009)

Dionysios Angelidis (University of Minnesota), Fotis Sotiropoulos (University of Minnesota)

A second order accurate method for discretizing the Navier-Stokes equations on 3D unstructured Cartesian grids is presented. Although the grid generator is based on the oct-tree hierarchical method, fully unstructured data-structure is adopted enabling robust calculations for incompressible flows, avoiding both the need of synchronization of the solution between different levels of refinement and usage of prolongation/restriction operators. The current solver implements a hybrid staggered/non-staggered grid layout, employing the implicit fractional step method to satisfy the continuity equation. The pressure-Poisson equation is discretized by using a novel second order fully implicit scheme for unstructured Cartesian grids and solved using an efficient Krylov subspace solver. The momentum equation is also discretized with second order accuracy and the high performance Newton-Krylov method is used for integrating them in time. Neumann and Dirichlet conditions are used to validate the Poisson solver against analytical functions and grid refinement results to a significant reduction of the solution error. The effectiveness of the fractional step method results in the stability of the overall algorithm and enables the performance of accurate multi-resolution real life simulations.

ORAL SESSIONS

Monday, November 25, 2013 Session G28: Waves II 9:44 AM–9:57 AM Room: Spirit of Pittsburgh Ballroom B/C	**Eulerian and Lagrangian effects of surface wave on turbulence underneath** (Abstract: G28.00009) Xin Guo (University of Minnesota); Lian Shen (University of Minnesota) Direct numerical simulation is performed to study the effects of surface wave on underlying turbulence. In the simulations, fully nonlinear kinematic and dynamic boundary conditions are applied at the free surface. The evolution of surface elevation is obtained by advancing the kinematic boundary condition with a Runge-Kutta scheme. In the vertical direction, grid is clustered towards the free surface to ensure the boundary layers of the free surface and surface wave are fully resolved. For spatial discretization, pseudo-spectral method is used in the horizontal directions, and second-order finite difference method is used in the vertical direction. The interaction of surface wave with underlying turbulence is carefully studied in both Eulerian and Lagrangian frames. In the Eulerian frame, turbulence statistics become wave-phase dependent due to the distortion of both the free surface and the periodic wave strain field. Budget of the Reynolds normal stresses is analyzed. In the Lagrangian frame, net effect of surface wave on turbulence is identified. It is found that the net wave effect is contributed by both the Stokes drift and the correlation between the wave field and the distorted turbulence field.
Monday, November 25, 2013 Session H11: Bubbles V: Rising Bubbles and Surface Interaction 12:14 PM–12:27 PM Room: 335	**Void fraction and bubble size in a simulated hydraulic jump (Abstract: H11.00009)** Adam Witt, (University of Minnesota); John Gulliver (University of Minnesota); Lian Shen (University of Minnesota) Two- and three-dimensional numerical simulations of a hydraulic jump are carried out with the open source software package OpenFOAM using a Volume of Fluid numerical method and a realizable k-ε turbulence model. Time-averaged air-water properties are obtained over a 15 second sampling time. Void fraction profiles show good agreement with experimental values in the turbulent shear layer. Sauter mean diameter approaches experimental results in the turbulent shear layer, while showing grid dependence down to a uniform computational cell size of 0.625 mm. Three-dimensional results show a minor improvement in the prediction of entrained air compared to two-dimensional results at a multiple of 341 in increased computational time for the chosen grid. Relative error in bubble diameter is similar between two- and three-dimensional simulations. The results indicate a Volume of Fluid, realizable k-ε numerical model accurately predicts the void fraction profile when the Sauter mean diameter to grid size ratio surpasses 8.
Monday, November 25, 2013 Session L18: Biofluids: General V 4:01 PM–4:14 PM Room: 306/307	**3D flow investigation near the denticles of biomimetic shark skin model using Digital In-line Holographic Microscopy** (Abstract: L18.00003) Mostafa Toloui (University of Minnesota); Jiarong Hong (University of Minnesota) It has been hypothesised that the complex microscopic denticles on a shark skin reduce the total drag for a swimming shark. However, the fundamental mechanism of this hydrodynamic function is not fully understood due to the inability to reproduce the complex shark surface and resolve the detailed flow around the skin denticles. Here we report a preliminary experiment using a 3D printed transparent rough surface replicating the morphological features of real shark skin. The model skin consists of closely-packed denticles of 2 mm in scale, i.e. $\sim$ 10 times of the real size. Particle image velocimetry based on digital in-line holography is employed to measure 3D flow structures. To reduce optical abberration and enable imaging around the denticles, we use a fluid medium that has the same optical refractive index as that of the skin model. The experiment is conducted in 2''x2'' square channel at a moderate Re number matching the general flow around a cruising shark. Several samples of the 3D velocity field amid and above the denticles are obtained. The follow-up research envisions a large dataset of these samples over the rigid/deformable model operated in stationary/undulating mode to ellucidate the dominant flow structures generated by the denticals.

ORAL SESSIONS

MONDAY, NOVEMBER 25, 2013

Monday, November 25, 2013
Session L3: Multiphase Flows VI
5:19 PM–5:32 PM
Room: 325

A computational model for large eddy simulation of dilute bubbly turbulent flows (Abstract: L3.00009)
Mohammad Hajit (University of Minnesota); Fotis Sotiropoulos (University of Minnesota)

A mathematical formulation of filtered equations for two phase bubbly flows based on two-fluid method is presented. To remove high frequencies (noise), we extracted the filtered form of the equations in curvilinear coordinates, converting the microscopic governing equations to macroscopic equations via spatial averaging of solution variables. The set of equations describing the hydrodynamics in a gas-liquid system can be solved effectively if the solution procedure is decoupled so that an efficient iterative scheme can be employed. We propose a formulation for dilute bubbly flows in which the equations are converted to a loosely-coupled form. The resulting mathematical model is based on five distinct sets of equations, namely mixture momentum balance, pressure Poisson equation, Boyle's law and momentum and mass balances of gas phase. This mathematical formulation provides an efficient numerical procedure for two-way coupling of bubbly flows at low gas holdups. The subgrid-scale modeling is based on dynamic procedure of Germano for both phases. The formulation is validated for a fully turbulent bubble column test by comparing to available experimental results.

TUESDAY, NOVEMBER 26, 2013

Tuesday, November 26, 2013
Session M13: Granular Flows IV: Mixing, Segregation and Separation
8:39 AM–8:52 AM
Room: 301

Segregation of Particles by Size and Density in Dense Sheared Flows: Gravity, Temperature Gradients, and Stress Partitioning (Abstract: M13.00004)
Danielle Tan (University of Minnesota); Kimberly Hill (University of Minnesota)

In sheared mixtures of different-sized (same density) particles modestly larger particles tend to go up (toward the free surface), and the smaller particles, down, commonly referred to as the ``Brazil-nut problem'' or ``kinetic sieving.'' If the larger particles are sufficiently denser than the smaller particles, the segregation reverses. Using theory and simulations, we have recently shown that the segregation fluxes among particles differing \textit{in size only} are driven by two effects: (1) the difference between the partitioning of kinetic and contact stresses among the species in the mixture and (2) a kinetic stress gradient. Specifically, the higher granular temperature of the smaller particles segregates them downward along a kinetic stress gradient toward lower temperatures, and larger particles upward. We adapt the theory to mixtures differing in both size and density and use simulations to show that when the larger particles are sufficiently dense, the theory captures the observed segregation reversal through a reversal in the relative granular temperature born by the two species. In other words, with increasing material density, the larger particles bear increasing fractions of the local kinetic stresses, and the segregation reverses as the larger particles bear a higher fraction than their local concentration in the mixture.

Tuesday, November 26, 2013
Session R1: Geophysical: General III - Open Channels and Sedimentation
1:31 PM–1:44 PM
Room: 323

Large-eddy simulation of density currents on inclined beds (Abstract: R1.00003)
Saurabh Chawdhary (University of Minnesota); Ali Khosronejad (University of Minnesota); George Christodoulou (National Technical University of Athens); Fotis Sotiropoulos (University of Minnesota)

Density currents are stratified flow in presence of density differential and gravity field. We carry out Large-Eddy Simulation (LES) to simulate the flow of a density current formed over sloped bed due to an incoming jet of heavy density salty water for two different cases of bed slope: (a) 5 degrees and (b) 15 degrees. The Reynolds and Richardson numbers based on inlet height and inlet velocity were (a) 1100 and 0.471, and (b) 2000 and 0.0355, respectively. The Schmidt number is set equal to 620, which corresponds to the value for salt-water. The computed results are compared with laboratory experiments in terms of overall shape of the heavy-density plume and its spreading rate and are shown to be in reasonable agreement. The instantaneous LES flow fields are further analyzed to gain novel insights into the rich dynamics of coherent vortical structures in the flow. The half-width of the plume is plotted as a function of downstream length and found to exhibit three different regions on a log scale, in agreement with previous experimental findings.

ORAL SESSIONS

TUESDAY, NOVEMBER 26, 2013

Tuesday, November 26, 2013 Session R1: Geophysical: General III - Open Channels and Sedimentation 1:44 PM–1:57 PM Room: 323	**Large-eddy simulation of coupled turbulence, free surface, and sand wave evolution in an open channel** (Abstract: R1.00004) Ali Khosronejad (University of Minnesota); Fotis Sotiropoulos (University of Minnesota) We develop and validate a coupled 3D numerical model for carrying out three-phase large-eddy simulations of turbulence, free-surface, and sand waves-bed morphodynamics under live bed conditions. We employ the Fluid-Structure Interaction Curvilinear Immersed Boundary (CURVIB) method of Khosronejad et al. (Adv. in Wat. Res.,2011). The LES is implemented in the context of the CURVIB method using wall modeling (Kang and Sotiropoulos, Adv. in Wat. Res.,2011). Free-surface motion is simulated by coupling the CURVIB method with a two-phase level set approach as in Kang and Sotiropoulos (Adv. in Wat. Res.,2012). Transport of bed load and suspended load sediments are combined in the non-equilibrium form of the Exner for the bed surface elevation, which evolves due to the spatio-temporally varying bed shear stress field induced by the turbulent flow. Simulations are carried out for the experiments of Venditti et al. (2005). It is shown that the model can accurately capture sand-wave initiation, growth, and migration processes observed in the experiment. The effects of free-surface on bed-form dynamics is also quantified by comparing the three-phase simulation results with two-phase simulations using a fixed rigid-lid as the free surface. This work is supported by NSF Grants EAR-0120914 and EAR-0738726, and National Cooperative Highway Research Program Grant NCHRP-HR 24-33.
Tuesday, November 26, 2013 Session R1: Geophysical: General III - Open Channels and Sedimentation 2:36 PM–2:49 PM Room: 323	**Suspension and transport of sediment under a plunging wave breaker** (Abstract: R1.00008) Xinhua Lu (Wuhan University); Xin Guo (University of Minnesota); Yi Liu (American Bureau of Shipping); Lian Shen (University of Minnesota) To understand the mechanism of suspension and transport of sediment under breaking water waves, we perform large-eddy simulations of a plunging breaker over seabed. The breaking water surface is captured by a coupled level-set and volume-of-fluid method. The mass exchange of sediment between the water region and the bottom is computed through the local upward erosion and downward deposition fluxes. The erosion flux is modeled based on the local shear stress at the bottom, and the deposition flux is estimated based on the sediment concentration near the bottom. We analyze in detail the instantaneous velocity and sediment concentration fields, the erosion and deposition fluxes near the bottom, as well as the bottom deformation under breaking waves. It is found that the sediment is mainly picked up from the bottom at the early stage of wave breaking, brought upwards, mixed by the turbulent motion, and then transported in the wave propagation direction by the current generated by the breaker. The wave breaking significantly enhances the horizontal transport of the sediment. It is also found that the air pocket entrained by the breaking wave plays an important role in the suspension, transport, and redistribution of sediment.

GALLERY OF FLUID MOTION POSTER AND VIDEO

Visualizing bubble dynamics in a simulated hydraulic jump (P041)
Adam Witt (University of Minnesota), John Gulliver (University of Minnesota), Lian Shen (University of Minnesota)

Bubble visualization in a simulated hydraulic jump (V033)
Adam Witt (University of Minnesota), John Gulliver (University of Minnesota), Lian Shen (University of Minnesota)