investigated by both experiments and computations. The present computation is based on the discrete singularity method. Furthermore, it is found that the equivalent diameter d_{e1} based on the hydraulic mean depth is the most adequate as a characteristic length scale to classify all the sloshing modes. The authors show an unified formula to predict the eigen frequencies, using the proposed modal classification and d_{e1}.

M-3E-3. ARRESTED TRACER MODEL FOR LONGITUDINAL DISPERSION IN RIVERS

Anton PURNAMA, *Sultan Qaboos University, Oman*, H. H. Al BARWANI, *Sultan Qaboos University, Oman,* A striking feature common to all measured data on longitudinal dispersion in rivers is the existence of a persistent skewness and, in particular, the collected tracer concentration distribution is characterized by an abrupt leading edge and an extended long tail. Another feature is that only a small fraction of the amount of tracer introduced into the river is frequently recovered at the most distance downstream. It is generally accepted that the long tail in the concentration distribution is caused by the tracer arrests by the stagnant zones found at the stream channel, such as sloughed banks and side channels, or behind protruding logs or boulders at the streambed. If more reliable predictive models are to be developed, it would seem important to compare predicted and observed values of the skewness. In the popular dead zone model, the tracer arrests and mixes are formulated using a simple mass exchange mechanism, and by comparing with the results obtained from the field tracer measurements, it is found that it fails to account for the persistence of skewness in observed data. Stream water movement into and out of the hyporheic zone, such as the saturated sediment beneath the stream channel, suggests that the arrested tracer might be transported along complicated pathways before eventually finding its way out, or perhaps not at all. In the arrested tracer model, we assume that the arrested tracer is mixed by a diffusion process in the semi-infinite stagnant zone, and therefore, the tracer spent a long period of time in the stagnant zone. Unlike the dead zone model, the arrested tracer model prediction is not characterized by decreasing the value of skewness. At large times, the model predicts a constant value of skewness, in agreement with the observed data collected.

M-3E-4. COMPUTATIONAL MODELLING OF THE IMPACT OF 2004 TSUNAMI ON THE CITY OF HAMBANTOTA IN SRI LANKA

J. J. WIJETUNGE, *Department of Civil Engineering, University of Peradeniya, Sri Lanka,* On 26th December 2004, coastal belts of Sri Lanka as well as several other countries bordering the Indian Ocean suffered enormous loss of life and damage to property owing to the tsunami unleashed by the third largest earthquake ever recorded. In order to mitigate potential loss of lives from a similar event in the future we need to provide advance warning of an approaching tsunami and then quickly evacuate vulnerable coastal communities to safer areas. Clearly, such evacuation planning requires prior information about vulnerable localities as well as areas that are deemed safe. The information necessary for this purpose is usually obtained through the development of tsunami hazard zonation maps which provide a graphical presentation of the spatial variation of the intensity of the probable depth of inundation and flow velocity across the areas of interest. Accordingly, the present paper outlines the numerical modelling of tsunami propagation and inundation carried out by employing non-linear shallow water equations to develop a high-resolution tsunami hazard map, as a case study, for the city of Hambantota on the south coast of Sri Lanka, which was devastated by the 2004 tsunami. The results give the spatial distribution of the maximum values of the depth of inundation as well as the flow velocities due to an event similar to the 2004 tsunami, which may be considered as a worst-case scenario in the absence of detailed probabilistic assessments of the tsunami threat for Sri Lanka. The model simulations confirm that the sand dunes, where present with sufficient elevation, have helped protect the settlements in their shadow from direct tsunami attack whilst comparatively vast extents of the salterns have acted as sinks to absorb and spread the flood water. The computed tsunami arrival times for the shoreline of Hambantota are also compared with eyewitness accounts.

16:30 ~ 17:50 (Room106)

Multiphase and Particle-Laden Flows (III)

Session Chair : Prof. N. Huang, Lanzhou Univ/China

M-3F-1. NUMERCAL SOLUTION OF THE CAVITATION OVER AXISYMMETRIC BODIES USING THE BOUNDARY ELEMENT METHOD BASED ON POTENTIAL

I. RASHIDI, M. PASANDIDE, N. GHAFOORIANFAR, M. MANSOUR, *Ferdowsi University of Mashhad, Iran,* Cavitation is recognized as an inadvisable problem in most phenomena, but in some circumstances, cavitation is remarked as a beneficial problem. The most important example is the submerged projectiles, in which cavitation is desired because of intense decrease in drag force. The dimensionless parameter which is represented for introducing cavitation is the cavitation number (σ). If bodies move with relatively high velocities inside fluids, cavitation starts at a point in which its local pressure reaches fluid vapor pressure. In low velocities or in high cavitation numbers, cavity is closed over the body and is called partial cavitation. With increase in velocity and decrease in cavitation number, cavity grows and covers all the body, which is called supercavitation. In 1993, Fine and Kinnas devised a nonlinear Boundary Element Method (BEM) based on potential elements for solving partial cavitation flow over a hydrofoil. Partial cavitation flow over torpedoes was conducted by Uhlman et al, using BEM method, and source and dipole distribution over body surface and cavity in 2003. Governing equation on the field of the flow is the Laplace equation. In this method cavitation will be modeled, by means of Green's third identity integral. This equation states that the potential flow on any surface can be shown by means of the ring distribution of sources and dipoles. For this purpose, the rings of the sources are distributed on the cavity surface, and also the rings of the dipoles are distributed on the body and the cavity surface. Applying Bernoulli equation, the relation between the total velocity on the cavity surface, and the cavitation number can be obtained which is called the dynamically boundary condition. The kinematic boundary condition states that the flow does not have any vertical component on the body and the cavity surfaces. In boundary element method (BEM) based on potential, the body and the cavity surfaces are respectively estimated by N_b and N_c number of the elements, which totally form N elements on the aforementioned surfaces. By discretization the governing equation and applying it on the surfaces of the body and the cavity, N number of the algebraic equation is obtained. The unknowns include: N_b number of dipole strengths on the body surface, N_c number of source strengths on the cavity surface, and a cavitation number. Therefore, the numbers of the unknowns are N+1, which is one more than the number of the equations. In order to resolve this problem and also solving the system of equations, an auxiliary equation is needed. To obtain this equation, the definition which states that the algebraic sum of the sources powers on the cavity surface must be equal to zero, is used. . The high velocity and also proper accuracy in calculating the geometry of the cavity are considerable advantages of this method.

M-3F-2. SOLID-LIQUID 2 PHASE HELICAL FLOW THROUGH A SLIM HOLE ANNULUS WITH ROTATING INNER CYLINDER

S. M. HAN, *Sungkyunkwan University, Korea,* N. S. WOO, *Sungkyunkwan University, Korea*, Y. K. HWANG, *Sungkyunkwan University, Korea*, Y. J. KIM, *KIGAM, Korea,* An experimental and numerical investigation was carried out to study solid-liquid mixture upward hydraulic transport of solid particles in a vertical and inclined annulus with rotating inner cylinder. Lift forces acting on a fluidized particle plays a central role in many importance applications, such as the removal of drill cuttings in horizontal drill holes, sand transport in fractured reservoirs, sediment transport and cleaning of particles from surfaces, etc. In this study a clear acrylic pipe was used in order to observe the movement of solid particles. Annular velocities varied from 0.4 to 1.2 m/s. Effect of annulus inclination and drill pipe rotation on the carrying capacity of drilling fluid, particle rising velocity, and pressure drop in the slim hole annulus have been measured for fully developed flows of water and of aqueous solutions. For higher particle volume concentration, the hydraulic pressure drop of mixture flow increases due to the friction between the wall and solids or among solids.

M-3F-3. OPTICAL MEASUREMENT OF VOID FRACTION AND FLOW PATTERNS OF GAS-LIQUID TWO-PHASE FLOW IN A MICROCHANNEL

H. IDE, R. KIMURA, M. KURAUCHI, *Kagoshima University, Japan,* M. KAWAJI, *University of Toronto, Canada,* An optical measurement system was developed to investigate gas-liquid two-phase flow characteristics in a circular microchannel of 100 μm diameter. By the comparison between optical signals obtained by multiple optical fiber probes and video images, mean void fraction was decided successfully. The time-averaged void fraction could be obtained from the time fraction by the passage of gas and liquid phases. These void data were obtained using a T-junction with the same internal diameter as the microchannel but the lengths of the gas and liquid injection lines between the T-junction and flow control valves were quite different in the present experimental conditions of Case 1 and Case 2. The presence of a large compressible gas volume upstream of the T-junction had a significant effect on the two-phase flow characteristics in the microchannel typified by the void fraction data. The effects of the threshold values of optical signals and flow patterns on the mean void fraction were also investigated.

M-3F-4. EULERIAN-LAGRANGIAN 3-D SIMULATION OF UNSTEADY GAS-LIQUID FLOW IN A RECTANGULAR BUBBLE COLUMN

A. FARZPOURMACHIANI, *K. N. Toosi University of Technology, Iran,* M. SHAMS, *K. N. Toosi University of Technology, Iran,* A. SHADARAM, *K. N. Toosi University of Technology, Iran,* R. EBRAHIMI, *K. N. Toosi University of Technology, Iran,* F. AZIDEHAK, *Material and Energy Research Center, Iran,* This work discusses the development of a three dimensional Eulerian-Lagrangian simulations of unsteady two-phase gas-liquid flow in a rectangular bubble column. The numerical simulations were carried out using a rectangular (0.2 m width × 4.5 m height × 0.05 m depth) bubble column. The superficial air velocity is 0.14 m/s and bubble sizes considered as 5 mm. The sparger through which gas was introduced in to the column was modeled as the area covered by the sparging holes (18×6 mm). The drag,lift, gravity, buoyancy, virtual mass and pressure gradient forces acting on a bubble rising in a liquid is considered. A two-way momentum coupling between the phases is considered. Several alternatives have been proposed to estimate the effective viscosity of the turbulent liquid phase in gas-liquid flow. Simulation of unsteady flow is a time consuming work therefore with assuming that flow is steady, standard κ-ε, RNGκ-ε, κ-ω and eddy viscosity transport models are considered to calculate flow properties. In comparison with experimental data, the standard κ-ε model of turbulence appears to perform satisfactorily. Therefore this model is selected for unsteady simulation. By solving the standard κ-ε model of turbulence for unsteady flow, the simulation results are compared with experimental data and shown that, the numerical solution results and experimental data are close to each other.

16:30 ~ 17:50 (Room107-108)

Computational Fluid Dynamics (III)

Session Chair : Dr. Nanoj Nair, IIT Kanpur/India

M-3G-1. COMUPTING FLOWS ON HIGHLY NONSMOOTH STAGGERD GRIDS

A. RABIEE, M. M. ALISHAHI, H. EMDAD, A. N. ZIAEI, *Shiraz University, Iran,* A method for computing the fluid flow in complex geometries using highly non-smooth and non-orthogonal staggered grid is presented. The pressure and the physical tangential velocity components are used as dependent variables in momentum equations. In this method, to reduce the sensitivity of the curvature terms in momentum equations to rapid change of the coordinate line orientation, the curvature terms are only computed using Cartesian velocity vectors. The method is then used to solve some fairly complicated 2-D and 3-D flow field using highly non-smooth grids. The method accuracy on rough grids was found to be high and the results showed good agreement with previous experimental and numerical results.

M-3G-2. A KIND OF ADAPTIVE ARBITRARY LAGRANGIAN-EULERIAN METHOD FOR THE COMPRESSIBLE EULER EQUATIONS

Y. J. WANG, *College of Science, Nanjing University of Aeronautics and Astronautics, China*, N. ZHAO, *College of Aerospace Engineering, Nanjing University of Aeronautics and Astronautics, China,* C. W. WANG and D. H. WANG, *College of Science, Nanjing University of Aeronautics and Astronautics, China*, Most of finite volume schemes in Arbitrary Lagrangian-Eulerian (ALE) method are constructed on the staggered mesh, where the momentum is defined at the nodes and the other variables (density, pressure and specific internal energy) are cell-centered. However, this kind of scheme must use a cell-centered remapping algorithm twice which is very inefficient. Furthermore, there is inconsistent treatment of the kinetic and internal energies [1]. Recently, a new class of cell-centered Lagrangian scheme for two dimensional compressible flow problems has been proposed in [2]. The main new feature of the algorithm is the introduction of four pressures on each edge, two for each node on each side of the edge. This scheme is only first order accurate. In this paper, a second-order cell-centered conservative ENO Lagrangian scheme is constructed by using an ENO-type approach to extend the spatial second order accuracy. Time discretization is based on a second order Runge-Kutta scheme. Combining a conservative interpolation (remapping) method [3,4] with the second order Lagrangian scheme, a kind of cell-centered high order ALE methods can be gotten. Some numerical experiments are made with this method. All results show that our method is effective and second order accuracy. At last, in order to further increase the resolution of shock waves, we have coupled an adaptive moving mesh method [5] with the ALE method. Numerical experiments are also presented to valid the performance of the proposed method.

M-3G-3. OPTIMAL AIRFOIL SHAPE DESIGN AT LOW REYNOLDS NUMBERS

SRINATH D. N., MITTAL. S., *Department of Aerospace Engineering, Indian Institute of Technology, India,* Significant literature exists for airfoils at high Reynolds numbers, but very little data exists at low *Re*. It is well known that airfoils designed for high *Re* operations are not suitable at low *Re*. Not much is known on the airfoil shapes that would have an optimum performance at low *Re*. A continuous adjoint based optimization method in the framework of a stabilized finite element formulation (SUPG/PSPG) is used here. The L-BFGS algorithm is used as the optimizer. The objective is to design high-lift producing airfoils at low *Re* in steady flows. A NURBS curve is used to parametrize the airfoil surface. The y - coordinates of the control points are used as the design variables. A mixed finite element mesh with structured triangular elements close to the surface and unstructured elements in the rest of the computational domain is employed for the computations. NACA0012 airfoil is used as the initial guess in the optimization cycle. It is shown that the process of determining the optimum is very dependent on how the objective function is defined. The flow is assumed to be steady and three different objective functions are studied at *Re*=10 and 500. They are 1) to maximize lift, 2) to maximize lift to drag ratio and 3) to maximize lift and to minimize drag simultaneously. For steady flows the final shapes obtained are seen to vary in performance with respect to the objective function. The final design is therefore seen to be dependent on how the objective function is defined. Computations are also carried out in the framework of unsteady flows is carried out at *Re*=500. The final shape obtained is seen to be different, with a better performance, from the one obtained assuming steady flow. Further studies are being carried out using different objective functions at different angles of attack and Reynolds numbers. The effect of using unsteady flow on the design process will be carried out.

M-3G-4. VISCOUS FLOW AROUND A PAIR OF SQUARE CYLINDER IN HORIZONTAL CHANNEL

V. ESFAHANIAN, *Department of Mechanical Engineering, Faculty of Engineering, University of Tehran, Iran*, B. KARAMI, B. BAGHAPOUR, M. A. MEHRABADI, H. Dashtaki HESARI, *Vehicle, Fuel and Environment Research Institute, University of Tehran, Iran,* Over the years, the unsteady flow around bluff bodies has been studied extensively because of its numerous engineering applications. Periodic vortex shedding patterns and fluctuating velocity fields behind the bluff bodies can cause structural damage as a result of periodic surface loading which shortens the life of the structure and increases the acoustic noise and the drag. Several practical configurations involve two or more bluff bodies in close proximity such as in the designs for heat exchangers, offshore structures and sea-bed pipelines in both air and water flow. Flows over bluff bodies are important in many engineering applications, especially in flows around bridges, buildings, marine risers. Investigation of three-dimensional laminar incompressible flow structure around a pair of square cylinders in a horizontal channel in presence of an incident linear velocity profile is the main purpose of this study. The flow field is considered for a separation distance between the wall and the cylinders wall-side face at 0.5 times the height of the cylinder. The effect of body-to-body on vortex formation is studied in three dimensions for different Reynolds numbers, 200, 300, and 400 (based on the height of the cylinder D and the incident stream U at the centerline of the cylinders). The unsteady Navier-Stokes equations are solved numerically through a second-order finite-volume scheme over a semi-staggered grid arrangement with arbitrary hexahedral shape. A SIMPLE scheme is employed for the convective terms. The diffusive terms are discretized in implicit manner while the convective terms are discretized in fully explicit manner. Finally, the results are compared for three different Reynolds numbers and three dimensional effects are shown for an arbitrary section of the three-dimensional channel. As a result in this paper, the fluid structure has no vortex generation between cylinders in small body-to-body distance. Moreover, vortex shedding appears behind the cylinders as Reynolds number increases.