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.