University of Tehran, Iran, S. GHADER, *Department of Space Physics, Institute of Geophysics, University of Tehran, Iran*, M. SHAHSAVARI, *Department of Space Physics, Institute of Geophysics, University of Tehran, Iran,* A two dimensional fully nonlinear numerical model is presented for simulation of internal gravity waves generated by a buoyancy forcing. The governing equations are written in terms of vorticity and density as prognostic variables, and stream function as a diagnostic variable. A Cartesian geometry with periodic boundary condition in horizontal direction and free-slip boundary conditions at upper and lower boundaries is utilized to perform the computations. A three level leapfrog time stepping method is used to advance the equations in time and the second-order centered finite difference scheme is applied to spatial differencing of the governing equations. Numerical results are presented for non-hydrostatic internal gravity waves. Numerical results are compared with some existing experimental observations showing that the flow structure in formation of shear layers (5-7 layers) is similar. It also appears that only the outflow due to either density point sources (numerical) or plumes (experimental) is important in formation of the internal waves that forms the layers.

M-1E-4. NONLINEAR BI-CHROMATIC WAVE-GROUP EVOLUTION DESCRIBED BY THE AB-EQUATION

MASHURI, *Institut Teknologi Bandung, Indonesia*, L. S. LIAM, *University of Twente, The Netherlands*, ANDONOWATI, *Institut Teknologi Bandung, Indonesia,* The evolution of finite amplitude water surface waves is known to be dominated by an interplay between nonlinear and dispersive effects. Although this is true for any wave-field, these effects are most clearly visible in wave-groups through the deformation of the envelopes. Nevertheless, only very few special wave-groups are known that have an analytic approximate description; maybe the best known one is the Soliton on Finite Background. Here we will study the down-stream evolution of a Bi-Chromatic wave-group using the recently derived AB equation. Although there seem to be no analytic approximations for this case, there are accurate measurements of experiments in large hydrodynamic wave-tanks. Comparison between the evolution described by the AB equation and the experimental results show promising results both qualitatively and quantitatively.

11:00 ~ 12:20 (Room106)

Multiphase and Particle-Laden Flows (I)

Session Chair : Prof. X. J. Zheng, Lanzhou Univ/China

M-1F-1. REMOVAL OF SEDIMENTS DEPOSITED ON RESERVOIR BEDS UTILIZING SIPHONAGE –SMALL SCALE TEST AND ANALYSIS

M. SADATOMI, A. KAWAHARA, T. TAKATA, *Kumamoto University, Japan*, In water reservoirs for city water and basins for settling sand and clay in hydraulic power stations, sand and mud etc. deposit on their beds, resulting water capacity reduction and water pollution. In order to remove such sediments efficiently, "A New Sediments Removal System Utilizing Siphonage" has been invented by Prof. M. Sadatomi. In the system, siphonage is utilized in order to minimize power consumption, and the suction port of the siphon can slide down automatically for the effective suction of sediments regardless of remaining sediments. As the first step of its development, a small scale apparatus having a siphon pipe of 20 mm I.D and the maximum level difference of H = 0.95 m from reservoir surface to siphon exit was constructed, and experiments have been conducted using the apparatus and 1, 2 and 4 mm O.D. ceramics spherical particles. In the experiment, the present system worked very well, and the data on the discharge rates of solid particles and water, Q_S and Q_L, and the solid particles volume fraction in siphon, α_S were obtained. The results showed that Q_L and Q_S data increased with the level difference, and with increasing of particle size, Q_L increased while Q_S decreased, and thus the ratio of Q_S/Q_L decreased. α_S data increased with decreasing of the particle size. In addition to the experiments, a mathematical model based on a one-dimensional steady state energy conservation equation has been proposed to predict the performance of the present system in order to find an optimum design method of the system in practical uses. The calculated results by the model agree well with the data for 2 and 4 mm particles in H > 0.6 m. For 1 mm particles, however, the agreement is not enough probably due to the periodical particles stoppages.

M-1F-2. MULTIGRID FICTITIOUS BOUNDARY AND FINITE ELEMENT METHOD FOR LIQUID-SOLID TWO PHASE FLOWS

D. C. WAN, *State Key Laboratory of Ocean Engineering, School of Naval Architecture, Ocean and Civil Engineering, Shanghai Jiao Tong University, China,* Direct numerical simulation of solid-liquid two phase flows with large number of moving particles is a very challenging task. The rigid particles are moved by Newton's laws under the action of hydrodynamic forces computed from the numerical solution of the incompressible viscous fluid equations. On the other hand, the fluid fields and domain are disturbed and changed simultaneously due to the motion of the particles. It is crucial that in the practical cases in which many moving particles often exist in fluids, the complex interaction between fluid and particles as well as the collision between particles put a great confrontation to any numerical schemes adopted. In this paper, an explicit multigrid fictitious boundary method (MFBM) coupled with finite element method to simulate the liquid-solid two phase flows with large number of moving particles is presented. The MFBM is based on a multigrid FEM background mesh and starts with a coarse mesh which may contain already many of the geometrical fine-scale details, and employs a (rough) boundary parameterization which sufficiently describes all large-scale structures with regard to the boundary conditions. Then, all fine-scale features are treated as interior objects such that the corresponding components in all matrices and vectors are unknown degrees of freedom which are implicitly incorporated into all iterative solution steps. The main advantage of the MFBM is that the solid particles, which are allowed to have different shape and size, can move freely through the computational mesh for the fluid part which has not to change in time. This MFBM approach can be easily incorporated into almost all CFD codes without the need for additional background meshes for the particles or special interpolation procedures since it only requires changes in the treatment of Dirichlet boundary conditions. Further, in the MFBM, very different shapes and sizes of particles can be easily included; even coalesence and breakup mechanism are possible. Finally, since the presented method is based on simple extensions of standard Navier-Stokes solvers, the 3D case can also be quite straightforward to be fulfilled. In this paper, as an illustration, two numerical simulations of three big disks plunging into 2000 small particles of three different densities and sedimentation of 5,000 particles in a cavity by the MFBM-FEM are presented. The numerical examples show that the presented method provides a robust and efficient approach to simulate solid-liquid two phase flows with large number of moving particles.

M-1F-3. EFFECTS OF A MAGNETIC FIELD ON HEAT TRANSFER COEFFICIENT IN A HEAT EXCHANGER USING MAGNETIC FLUID

H. TSUBONE, *Ariake National College of Technology, Japan*, Y. NISHIMARU, *Mitsubishi Heavy Industries Ltd. Hiroshima Machinery Works, Japan*, Y. KOGA, *Ariake National College of Technology, Japan,* In recent years, magnetic fluid has been developed for a variety of new applications. However, there are not many practical applications for utilizing magnetic fluid. For the purposes of practical applications of magnetic fluid, the authors have proposed a new type of heat exchanger using magnetic fluid, which is capable of controlling heat transfer by means of a magnetic field. Although a lot of study on heat transfer of magnetic fluid have been conducted by many researchers, the effect of location and strength of magnetic fields on heat transfer coefficients in vertical circular pipe are not clarified yet at present. The purpose of this study is to clarify the phenomenon experimentally based on the above mechanism. In this experiment, a water-based magnetic fluid was used as a working fluid under atmospheric pressure. The test pipe in the test section was circular with a 10.2 mm i.d. and 200 mm in length, made of brass. Different sets of permanent magnets were placed at the test section. Experimental data on temperature, heat transfer coefficient etc. were measured under steady state at different experimental conditions of heat flux (q=1.5×104-3.4×104 W/m2), liquid volumetric flux (jL=0.2-0.4 m/s), positions of the magnetic field for the test section (z=0 mm and 45 mm) and strengths of magnetic fields between the magnets at a center axis (H=0.0093 – 0.0277 MA/m). Then, experimental result on temperature, heat transfer coefficient etc. were analyzed for various parameters. Especially, effects of position and strength of the magnetic field on heat transfer coefficient in heat exchanger using magnetic fluid were demonstrated. As a result, from the present data, it was confirmed that heat transfer using a magnetic fluid in a heat exchanger can be controlled by location and strength of the magnetic field. In addition, the relationships between Nusselt number and Reynolds number were discussed and some experimental correlations on Nusselt number are proposed in this paper.

M-1F-4. A STUDY OF SEPARATION AND REATTACHMENT PROCESS IN GAS-LIQUID TWO PHASE FLOW

A. IIJIMA, *Tokyo University of Science, Japan,* T. TANABE, *Japan,* M. MOTOSUKE, *Tokyo University of Science, Japan,* S. HONAMI, *Tokyo*

University of Science, Japan, In Electro Chemical Machining process, generation of air bubbles and the flow separation complicate the flow behavior of electrolyte and decrease the shape accuracy of the product. It is important to understand the influence of air bubbles on the separated flow. In this study, the flow characteristics of gas-liquid two-phase flow with millimeter-scale bubbles and separation in a channel are examined. Bubbles are fed to the liquid phase from stainless-steel pipes. Flow separation is generated by a square rod. All the experiments are conducted at the bulk velocity, U_b, of 370 mm/s in the single-phase flow, corresponding to the Reynolds number of 2000 based on the square rod height and the bulk velocity. The void fraction α is set from 0 to 0.5 %. In order to compare the effect of the bubble diameter with the same void fraction, 3-pentanol ($C_5H_{11}OH$) of nearly 30 ppm is added as a surfactant. We have applied a PIV system with fluorescent particles to the measurement of the liquid flow. In order to capture the shape of bubbles, the projected shadow image technique is applied. For the case without the surfactant, bubble diameter takes the peak value at about 0.7 and 2.9 mm. On the other hand with the surfactant, it takes the peak value at about 0.7 mm within a small dispersion for all conditions. The streamwise velocity becomes higher in the core flow region and the flow reversal as the void fraction increases, while that in the separating shear layer become lower. For the case of the flow without surfactant, streamwise velocity fluctuation takes high value at the high void fraction. Also, transeverse velocity fluctuation is higher compared with in the single-phase flow. On the other hand, in the case with the surfactant, transverse fluctuation becomes lower as the void fraction increases.

11:00 ~ 12:20 (Room107-108)

Computational Fluid Dynamics (I)

Session Chair : Dr. S. K. Hong, ADD/Korea

M-1G-1. STABILITY ANALYSIS OF UNIFORM ANNULAR PASSAGES CONDUCTING INCOMPRESSIBLE LAMINAR FLOWS FOR SMALL AMPLITUDE OSCILLATORY ROTATION OF THE OUTER CYLINDER

H. YARJIABADI, *Department of Mechanical Engineering Buali-Sina University, Iran,* In this paper a computational method is developed involving the simultaneous integration of the Navier–Stokes and structural equations for the purpose of studying the stability of concentric annular passages conducting incompressible laminar flows. It is assumed that one side of the annulus, i.e. the centre-body, is fixed and the outer cylindrical duct is flexibly supported. The outer cylinder is displaced or rotated from its equilibrium position and is then released. In this situation, the fluid part of the problem is solved by an accurate method using a three-point backward implicit scheme, followed by a pseudo-time iteration using an artificial compressibility factor. The fluid equations are discretized in space based on a finite-difference formulation and primitive variables, for which stretched staggered grids are used. The resulting equations are cast in delta form and are solved using an Alternating Direction Implicit (ADI) scheme. The fluid forces acting on the vibrating cylinder are calculated from the integration of the unsteady pressure and shear stresses resulting from the unsteady primitive variables calculated. The equations of motion of the structure, subjected to the calculated fluid forces are solved using the Runge–Kutta scheme to obtain the displacement or rotational angle of the moving cylinder. The problem is solved for small amplitude motions, by means of the so-called mean position (MP) analysis. It is shown that, between translational and rotational motions of the outer cylinder, the most stable configuration is that of rotational motion. This study can be used for FIV and FSI analysis of the annular structures.

M-1G-2. SIMULATION OF FLOW AROUND A MOVING BODY USING UNSTRUCTURED CHIMERA GRID METHOD

S. Z. NI, X. ZHANG, G. W. HE, *Institute of Mechanics, CAS, China,* Flow around a moving body is ubiquitous in practical problems such as insect flight, fish swimming and leave falling. The Chimera (or overset) grid method is an efficient method for treating such problems. In this method, Grids for each sub-domain can be generated separately. Governing equations are also solved independently and the solutions of each sub-domain are coupled through the exchange of information across the interior boundaries. The unstructured grid can easily generate the mesh complex geometry and reduce the number of sub-domains significantly. The present method preserves the advantages of the Chimera grid method and the unstructured grid. A novel numerical scheme proposed recently by our research group is used to solve the incompressible N-S equations and a “hole-cutting” method is developed to determine the intergrid boundary in background mesh. The moving mesh control volume method is deployed to treat the moving sub-domain. For the moving control surface, the space conservation law (SCL) has to be satisfied. The SIMPLEC algorithm is used to couple the pressure with the velocity. A second-order upwind scheme is used for the discretization of convective term and the Crank-Nicholson scheme is used for the temporal advancement. The collocated grid arrangement is deployed and a Rhie-Chow interpolation is used to compute mass fluxes at faces to eliminate pressure oscillation. And the Schwarz method is followed to couple the solutions in different sub-domains. Flows around an oscillating circular cylinder and a hovering wing are simulated. The numerical results are in good agreement with other experimental and computational data in literature. Basic features of flow around these moving bodies are successfully captured. These numerical examples demonstrate the capability of our method in handling moving boundaries.

M-1G-3. NUMERICAL SIMULATION OF FLOW OVER PITCHING BODIES USING AN IMPLICIT REYNOLDS AVERAGED NAVIER STOKES SOLVER

Sharanappa V. SAJJAN, Vimala DUTTA, P. K. DUTTA, *Computational and Theoretical Fluid Dynamics Division National Aerospace Laboratories, India,* There is a great demand for efficient and accurate computation of unsteady aerodynamic loads for aeroelastic investigation of modern aircraft in the transonic regime. The availability of high speed computers and advances in numerical methods in recent years have led to the successful development of sophisticated aeroelastic simulation techniques. Recently, several aeroelastic applications have employed Euler and Reynolds-Averaged Navier-Stokes (RANS) solvers for flutter analysis using both frequency domain and time domain approaches. However, in order to be an integral part of a flutter analysis package, the Euler or RANS solver is to be evaluated through time accurate computations for flow past standard aeroelastic configurations. The present paper is an attempt towards demonstrating the capability of an implicit Reynolds-averaged Navier-Stokes solver (IMPRANS) for simulating unsteady compressible flows over pitching aerofoils and wings. The RANS solver used for obtaining the time-accurate solutions is based on an implicit finite volume nodal point spatial discretisation, wherein a control volume is formed by joining the centroids of the neighbouring cells around a nodal point in the computational domain. The algebraic eddy viscosity model due to Baldwin and Lomax is used for turbulence closure. Computations are presented for transonic flow past a pitching NACA 64A10 aerofoil and a pitching LANN wing. The aerodynamic coefficients as well as the mean and fundamental frequency pressure data for these two- and three-dimensional aero-elastic configurations at transonic Mach numbers are found to be in good agreement with the experimental data. While the mean surface pressure distribution compares very well for both the cases, the first harmonic shows only reasonable agreement over the outboard region of the LANN wing, possibly due to insufficient grid size. The present work thus demonstrates the capability of the solver to provide useful unsteady pressure data for aero-elastic analysis.

M-1G-4. FLOWS INDUCED IN A FLUID-FILLED COMPLIANT CYLINDER OSCILLATED BY AN EXTERNAL FLOW

M. YOKOYAMA, O. MOCHIZUKI, *Toyo University, Japan,* The objective of this research is to understand internal flow fields inside a fluid-filled compliant cylinder placed in a uniform flow. The inside flow induced by the deformation of a soft-matter object was investigated numerically. Since the pressure and shear stress determined by the flow fields act on the outer and inner surfaces of the membrane of the cylinder, the Navier-Stokes equation and equation of motion of the soft material were combined. The membrane consisted of units of a mass-spring-damper (MSD) system. The simulation was carried out in a 2-dimensional region. The initial cross section of the compliant cylinder was a circle and placed at the origin of the coordinates. The non-slip condition was satisfied at the wall. The mesh was regenerated every time steps to fit the deformed shape of the cylinder. The inner fluid was the same as that of the outer flow in this study. We showed the change in aspect ratio of the cross section and the velocity fluctuation due to the Kármán vortices. The AR oscillates with the natural frequency of the cylinder for a while after the beginning of the calculation because of the impulsive start. After the natural vibration settles, the rear wall vibrates with the shedding frequency of Karman's vortices. The amplitude of the vibration of the rear wall was approximately 0.2% of the initial diameter of the cylinder. The inner flow pattern during the natural vibration became the saddle point. On the other hand, the source-sink flow between the different portions at the rear wall inside the cylinder is induced by the excitation due to the shedding vortices. The direction of the flow becomes opposite due to the shedding frequency. This indicates the possibility to be applied the outer vibration to mixture of different liquids inside a compliant container.