pressures, but depart from each other after a few microseconds. In the atmospheric spray, the penetration depth versus time has a nearly constant slope at all times but in other cases, the slope of the curve depends on time.

M-2F-2. ON TURBULENCE MODULATION TERM IN A PARTICLE-LADEN TURBULENT CHANNEL FLOW

B. WANG, *Department of Engineering Mechanics, Tsinghua University, Beijing, China,* The closure problem of turbulence modulation terms is one of the key issues both in particle Lagrangian trajectory model and in Eulerian-Eulerian two-fluid model. It's difficult to measure these modulation terms in experiments. Therefore, examinations of closures are mostly based on numerical simulation methods. In this study, one-way coupling large eddy simulation is used to calculate the gas-phase turbulent channel flow, with a friction Reynolds number Re_τ =180. Three different kinds of particles: lycopodium (700 kg/m^3, 30μm), glass (2500 kg/m^3, 57μm) and copper particles (8900 kg/m^3, 80μm), which are heavier than gas-phase, are chosen to release into the fully developed turbulent flow field. All the individual particles are tracked by Lagrangian trajectory method, neglecting SGS turbulence fluctuating velocity on particle motion. The corresponding to statistics results are obtained by averaging data spatially and temporally. Based on the statistics of two-phase velocity fluctuation terms, two closure models proposed by Chen & Wood, Mostafa & Mongia, respecitively, widely applied in RANS, are examined. The streamwise normal-correlation term has the highest level, and the largest value happens at about y^+~15. The normal-correlation term in the wall-normal direction has the lowest level. As particles Stokes number increases, all the correlation terms decreases in the whole channel, because the particle inertia filtering function becomes very strong. Lagrangian integral time scale τ_{Li} is calculated to parameterize particle Stokes number, which is computed in a Lagrangian way assuming an exponential decay of the fluid velocity autocorrelation function. Particle Stokes number based on the Lagrangian integral time scale has largest value near the wall and decreases about one-order magnitude towards the channel central plane. It's shown that for the streamwise two-phase correlation term, both two models predict poorly, but Chen & Wood's model is better than Mostafa & Mongia's model. Both two models predict better in the wall-normal and spanwise direction than in the streamwise direction. In the near-wall region, y^+<10, both two models predict poorly in three directions for three kinds of particles, because the local equilibrium conditions between $<u_{fi}'u_{pj}'>$ and $<u_{fi}'u_{fj}'>$ fail in construction of the models.

M-2F-3. THE EFFECT OF AIR ENTRAINMENT ON THE PRESSURE DAMPING IN PIPELINE TRANSIENT FLOWS

T. S. LEE, *NUS, Singapore*, H. T. LOW, *NUS, Singapore*, D. T. NGUYEN, *NUS, Singapore,* Studies show that even a small amount of air entrainment in the flow can give significant effects to the fluid transient. In general, the first pressure peak with entrained air is found to be higher than that predicted by models with no air. The pressure transient damping with air entrainment is faster than the damping with no air entrainment. The damping of pressure surge is contributed by different sources and processes. From the corresponding numerical experiments it is noted that friction and any devices put into the system have a damping effect on the pressure waves owing to the hysteresis in the energy cycle. However, the damping produced by losses alone is small and is independent of the local surge pressure encountered. The transient shear stress has only a small damping effect on the pressure fluctuations. The thermal exchange between the gaseous phase and the surrounding liquid does not always explain the observed dissipation, and frictional dissipation does not appear to be mainly responsible for the oscillation damping. The remaining prime factor on pressure damping is the effect of air entrainment which is considered in the below investigation. The paper aims to investigate the effect air entrainment on the damping of pressure transients in pipeline flows. A set of experiments were carried out in a single pump pipeline system to test the fluid transient behaviour for the case of pump trip due to power failure. A variable wave speed model is used to simulate this transient flow which includes the effects of free gas in the fluid on the pressure transients in the pipeline. This model is solved numerically by using the method of characteristics. The experimental results were investigated together with numerical results using the variable wave speed model. The results show that the peak pressure varies significantly with increasing initial air void fraction. Moreover, the rate of pressure damping is higher caused by the increased effective bulk viscosity of the fluid as a result of the presence of air entrainment. This is due to the fact that the increase of initial air void fraction leads to the decrease of the wave speed. The reduction of the wave speed of the mixture directly causes changes in the strength of pressure oscillations. In further research, the effect of 2D flow should be taken into account to provide more accurate predictions of pressure transient in pipeline system.

M-2F-4. FLUID-ELASTIC INSTABILITY OF NUCLEAR STEAM GENERATOR U-TUBES IN TWO-PHASE FLOW

I.-C. CHU, H. J. CHUNG, Y. J. YOUN, *Korea Atomic Energy Research Institute, Korea,* In the present study, the critical velocity of fluid-elastic instability, damping ratio, and hydrodynamic mass were experimentally investigated, and the instability constant of Connors' equation was evaluated for a rotated square array U-tube bundle with p/d of 1.633 in air-water two-phase flow. In addition, the effects of primary side flow and anti-vibration bar (AVB) on the vibration displacement of U-tubes were examined. The vibration displacements of the U-tubes showed that the out-of-plane (OP) first mode vibration dominated over the in-plane (IP) mode vibrations. Fluid-elastic instability was clearly observed in the OP first mode vibration. The damping ratio of the present U-tubes was higher than the damping ratio of the cantilever tubes in the literature. The instability constant (K) of the Connors' equation was assessed with a simplified effective gap velocity based on the void fraction measurement and the flow visualization. The K was in the range of 6.5 ~ 10.5. Another set of experiments was carried out to examine the effect of AVBs on the vibration displacement of U-tubes for the void fraction of 70 - 95 %. The vibration displacements in the OP 1st mode were significantly decreased after the AVBs were installed. However, the effect of the AVBs on the IP 1st mode vibration was not that significant and different among the U-tubes. This might due to the difference in the gap and contact force between the U-tubes and AVBs.

14:50 ~ 16:10 (Room107-108)

Computational Fluid Dynamics (II)

Session Chair : Dr. M. Shigeta, Tohoku Univ/Japan

M-2G-1. MATHEMATICAL TREATMENT INVESTIGATION OF AXISYMMETRIC SUPERCAVITATING FLOW PARAMETERS, USING BOUNDARY INTEGRAL METHOD

R. SHAFAGHAT, *IUST, Iran*, S. M. HOSSEINALIPOUR, *IUST, Iran*, A. VAHEDGERMI, *IUST, Iran,* Supercavitation has become a hot topic due to its potential to significantly enhance the speed of undersea weapons, projectiles, and vehicles by enabling them to travel inside a vaporous or gas fed cavity through significant reduction of viscous drag. The boundary integral method (BIM) is formulated numerically for the problem of the unbounded potential flow past a supercavitating body of revolution (flat disk) placed perpendicular to the flow direction. In the analysis of potential flow past a supercavitating flat disk, a cavity closure model must be employed in order to make the mathematical formulation close and the solution unique. In the present study, we employ Riabouchinsky closure model. According to this model, an image body is introduced at a certain distance downstream of the real body to make the cavity closed. It should be noted that this closure model has been chosen as a compromise between simplicity and physical validity. An iterative scheme is employed to locate the cavity surface. Upon convergence, the exact boundary conditions are satisfied on the body-cavity boundary. For this work, a software pack, based on CFD code, is developed. The predictions of the software are compared with those generated by analytical solution and with experimental data. The predictions of software for supercavitating flat disk are seen to be excellent. Using the obtained data from software, we investigate the mathematical treatment of axisymmetric supercavitating flow parameters including drag coefficient of supercavitating flat disks, cavitation number and maximum cavity width for a wide range of disk diameters and cavity lengths. The main objective of this study is to propose appropriate mathematical functions describing the behavior of these parameters.

M-2G-2. LESS DISSIPATIVE CHARACTERISTIC SCHEME IN *X-Y-T* SPACE

R. TRIATMADJA, *Gadjah Mada University, Indonesia,* The Method of Characteristics has been widely used for solving free surface flows in *X-T* and *X-Y-T* spaces. Apart from the advantage of the method, the existing characteristic schemes are dissipative in both *X-T* and *X-Y-T* which sometime make the scheme not suitable for simulating wave propagation in significantly large area. In this paper an additional factor (plane) that reduces the dissipation of the characteristic scheme in *X-Y-T* is developed. The term representing the additional plane is a function of both $L/\Delta s$ and Courant number to keep the scheme stable. Hence, information on the level of discretization ($L/\Delta s$) at grid points is developed. The scheme was

tested to simulate linear wave oscillation in *X*, and *X-Y* oscillations. The results show that the scheme is significantly less dissipative especially in *X-Y* oscillation. For simulation along *X* direction only, the scheme shows significantly reduce dissipation when Courant number is less than 0.5. Simulations of more complex wave (two waves of different length) oscillations in *X-Y-T* space were conducted at low resolution. The longer wave is represented by 16 grid points whilst the shorter is represented only by 8 grid points. The results indicate that the scheme is superior when compare to the previous characteristic schemes. After even 20 oscillations of the longer wave (40 oscillations of the shorter wave), the group can still be easily recognized. On the other hand, the shorter wave dies out after 20 oscillations with the original quadratic interpolation scheme. The difference of the numerical wave speed between the longer wave and the shorter one is easily observed. It is concluded that the new interpolation scheme in *X-Y-T* space is convergence at appropriate value of *K* and *f*. It also improves the performance of the characteristic scheme in *X-Y-T* space.

M-2G-3. AN INVESTIGATION AND COMPARISON OF ROE UPWIND METHODS WITH CUSP CENTRAL DIFFERENCE SCHEMES

Mahmood P. FARD, *Ferdowsi University of Mashhad, Iran*, M. SALARI, M. MANSOOR, M. Malek JAFARIAN, *Birjand University, Iran,* In this paper, Euler equations have been solved for the internal compressible flows. In order to have such a solution, at first, the equations were written into the integrated form, and then in CUSP and Scalar schemes, by means of the central difference method, and in Roe scheme, by means of the upwind difference method, they were made into the discritized form. The main problem in the central difference method for solving these equations that occurs as a result of the shock phenomenon is instability and discontinuity of the solution, which is due to the misuse of the information related to the both sides of the shock. In order to solve this problem, we have to apply the artificial dissipations. However, the upwind methods based on the distribution of the flow information along the defined directions in the physical domain; do not have such a problem. In this paper, these schemes will be introduced, and the results of these methods will be compared and then the CUSP scheme will be amended and we will study the performance of central and upwind schemes in solving Euler and Navier-stoks equations. Since, the central difference schemes do not imply enough dissipation, the artificial dissipation, which has an important effect on the accuracy of the solutions, should be added to the equations. The second group is the upwind methods. These methods are based on the distribution of the flow information along the defined directions in the physical domain. So, these methods have a good coincidence with the physics of the flow information all over the field of the fluid flow. After representing Roe and CUSP schemes, we will compare them and their results in solving Euler equations in a nozzle. The results of these methods will be represented for flows with different Mach numbers. The results which have been obtained show that CUSP scheme for low supersonic flows, comparing with the other methods, leads to more accurate results to capture the shock. However, for the higher Mach numbers, it seems it needs some corrections. Thus the modified CUSP scheme which is based on the CUSP itself, reveals more accurate results the high Mach numbers in the vicinity of shock.

M-2G-4. NON-LINEAR MIXED-OBERBECK ELECRTOCONVEC TION IN A POORLY CONDUCTING FLUID THROUGH A VERTICAL CHANNEL

B. S. SHASHIKALA, *Department of Mathematics, Siddaganga Institute of Technology, India*, N. RUDRAIAH, *Centre for Advanced Studies in Fluid Mechanics, Department of Mathematics, Bangalore University, Bangalore,* The study of combined free and forced convection called mixed convection (MC) in a vertical channel has received considerable attention because of its wide range of applications from cooling of electronic devices, gas-cooled nuclear reactors to that of solar energy collectors. In contrast to MC due to variation of density with temperature discussed above there is another type of MC arising in a poorly conducting alloys like nickle-titanium(N_i-T_i), aluminum oxides and so on in a vertical channel due to variation of electrical conductivity, σ ,with temperature in the presence of an electric field, $\vec{E}$,called mixed electroconvection (MEC). The variations of σ with temperature releases the charges forming distribution of charge density, ρ_e. These charges in turn induce the electric field, $\vec{E}_i$ called thermal or induced electric field. In addition there may be an applied electric field, $\vec{E}_a$, due to embedded electrodes of different potentials at the boundaries. The total electric field $\vec{E}(=\vec{E}_i+\vec{E}_a)$ in a poorly conducting fluid produces a current, which acts as sensing. In addition this $\vec{E}$ together with ρ_e produces an electric force $\rho_e\vec{E}$ which acts as actuation. These two properties are the two important properties required a material to be a smart material. Therefore the control of MEC plays a significant role to synthesize smart materials for practical use in science, engineering and technology. Is spite of these importances much attention has not been given to the study of MEC in a vertical channel and is studied in this paper. The nonlinear-coupled momentum and energy equations are solved numerically using finite difference technique and analytically using regular perturbation method with B_r as the perturbation parameter. The analytical and numerical solution are in good agreement. These results in the presence of total electric field, viscous and Joulean dissipations obtained are useful in the effective control of heat transfer in many industrial problems.

16:30 ~ 17:50 (Room101)

Flows in Porous Media (II)

Session Chair : Prof. J. Zhou, CAS/China

M-3A-1. REQUIRED HEAD FOR LOOSE-FINE MATERIAL FLUIDIZER SYSTEM

R. TRIATMADJA, *Gadjah Mada University, Indonesia*, A. THAHA, *Hasanuddin University, Indonesia*, N. YUWONO, *Gadjah Mada University, Indonesia*, NIZAM, *Gadjah Mada University, Indonesia,* Maintenance dredging and channel protections using coastal structures may solve the problem of sedimentation although they may not always be cost efficient. An alternative approach namely fluidization technique may be used to fluidize the sediment and flush the sediment out in to the sea. In this paper, the required head for full fluidization stage is studied using one dimensional, two dimensional and three dimensional fluidizer system models. The one dimensional model is used to study the initial and full fluidization in a tube. Two dimensional models are utilized to study the required head for full fluidization at various condition of sediment height (d), water depth (h), diameter of the perforation holes (D_f), and the distance of the perforation holes (a). Finally three dimensional full scale models are constructed to study the required head for full fluidization. The scale model is designed so that the scale effect of the model is minimized. The study shows that the distance of perforation holes and the diameter of the perforation holes play important roles. The longer is the distance between the perforation holes the less is the sharing of the jets and hence, higher required pressure head. Smaller perforation holes require higher head to maintain effective jets' strength. The required head for full fluidizations h_f/d_b is found to be strongly correlated with $D_f / (a.d_b)^{0.5}$ *a.* The head requirement at full fluidization may also be shown as a function of D_f/a. When plotted together with data from previous results it is apparent that all the data have similar trend where h_f/d_b is decreasing with increasing D_f/a. The theoretical line agrees quite well with the present experiment. Finally it may be concluded that the required head for full fluidization may be represented by the following equation: $\frac{h_f}{d_b} = p\frac{\sqrt{a\,d_b}}{D_f}$, where *p* is approximately 0.23.

M-3A-2. DOUBLE DIFFUSIVE CONVECTION IN A NON-NEWTONIAN FLUID SATURATED POROUS LAYER WITH THROUGHFLOW

S. SURESHKUMAR, *Department of Mathematics, Siddaganga Institute of Technology, India,* I. S. SHIVAKUMARA, *Department of Mathematics, Bangalore University, India,* The effects of quadratic drag and vertical throughflow on the onset of double diffusive convection in a non-Newtonian fluid saturated horizontal porous layer are investigated. The study finds its applications in chemical engineering, geothermal systems, enhanced recovery of petroleum reservoirs, underground spreading of chemical wastes, sea bed hydrodynamics and more importantly in the directional solidification of alloys, where a mushy zone exists which is regarded as a porous layer with double diffusive origin. Many practical problems cited above involve non-Newtonian fluid flow through porous media which is based on a generalized Darcy equation. There exist many different types of non-Newtonian fluids. However, some oil sand contains waxy crude oil at shallow depths of the reservoirs which are considered to be viscoelastic fluids. In such situations, a viscoelastic model of a fluid will be more realistic than inelastic non- Newtonian fluids. Also, many geophysical and technological applications involve non-isothermal flow of fluids through porous media, called throughflow. A modified Forchheimer-extended Darcy model which takes inertia into account and viscoelastic effects is employed to describe the flow in a porous medium. The boundaries are considered to be impermeable and perfect conductors of heat and solute concentration. Conditions for the occurrence of stationary and oscillatory convection are obtained analytically using the Galerkin technique. It is shown that oscillatory convection occurs even if Λ, the