blade and that with no diffuser blade, the flow vectors is fluctuated with changing the relative positions of impeller blades and diffuser blades, that is, the flow in with diffuser blade impeller has a strong unsteady flow. Furthermore, by comparing the results between EFD (LDV measurement) and CFD (CFX-code simulation) analysis, we discuss the relation between the impeller performances and flow distribution in the impeller in detail.

M-1D-2. PREDICTION OF PIG MOTION THROUGH NATURAL GAS PIPELINES

S. ZIAEI-RAD, M. D. EMAMI, *Isfahan University of Technology, Iran,* M. RAFEEYAN, *Yazd University, Iran,* Pipeline Inspection Gauge (PIG) is a device which is widely used in the pipeline transportation of fluids. PIG can perform a number of tasks including cleaning debris, removal of residual, and gauging the internal bore of the pipeline. Failure of the pipeline or its performance deterioration may be due to different reasons, such as the deflection of the pipes, corrosion, the increase in the pipe roughness and the obstruction of the flow area. Running PIG inside the pipeline is an effective measure to prevent these unwanted situations. PIG is also used to monitor the physical conditions of the pipeline. The performance of the PIG depends on its kinematics characteristics, namely, its velocity and acceleration. An estimation of these parameters is essential in adopting the appropriate PIG for the pipeline service. A literature survey reveals few papers dealing with the dynamic analysis of PIGs in pipelines. Most of the research results are commercially based or field experience. There are some papers that concentrate on the motion of PIGs and their dynamics in pipelines. Transient PIG motion through gas and liquid pipelines was studied assuming a plane, straight pipe. This paper presents a method for calculating the PIG motion in pipelines. The PIG speed may control through the amount of bypass flow across its body. The dynamic behavior of the PIG depends on the pressure difference across its body and the bypass flow through it. The system dynamics includes: dynamics of driving gas flow behind the PIG, dynamics of expelled gas in front of the PIG, dynamics of bypass flow, and dynamics of the PIG.

M-1D-3. EXPERIMENTAL STUDIES ON HEAT TRANSFER ENHANCEMENT OF TURBULENT FLOW THROUGH A CIRCULAR TUBE WITH WAVY TWISTED TAPE INSERTS

S. EIAMSA-ARD, *MUT, Thailand,* C. THIANPONG, *KMITL, Thailand,* R. CHAICHOMPOO, *MUT, Thailand,* P. EIAMSA-ARD, *MUT, Thailand,* P. NIVESRANGSAN, *MUT, Thailand,* P. PROMVONGE, *KMITL, Thailand,* Experimental investigations of turbulent heat transfer and friction factor characteristics in a tube fitted with wavy twisted tape have been made. In the experiments, the twisted tape with wavy edge is inserted in a uniform heat flux tube with a view to generating swirl flow that assists to increase the heat transfer rate of the tube. Tube with wavy twisted tapes having different twist ratios (y/w) inserted into a horizontally positioned plain tube has an inner diameter of 48 mm and a length of 1.25 m. The twist ratios (y/w) of the tapes are 4.0, 5.0, and 6.0, respectively. The flow rate of the tube is considered in terms of Reynolds number between 4,000 and 20,000. The experimental data obtained are compared with those from plain tube published data. The experimental results show that the mean heat transfer enhancement of the tube fitted with wavy twisted tape of y/w=4.0 are around 140% and the heat transfer coefficient increases with the decrease of twist ratio (y/w). The empirical correlations developed in terms of twist ratio (y/w) and Reynolds number, are well fitting the experimental data within ±10% for both Nusselt number and friction factor.

M-1D-4. A STUDY ON THE FLOW DISTRIBUTION TO THE CHANNEL IN THE PLATE HEAT EXCHANGER

Z. H. JIN, G. T. PARK, D. S. HEO, S. H. CHOI, H. S. CHUNG and H. M. JEONG, *Gyeongsang National University, Korea,* Plate heat exchanger (PHE) is an important part of condenser and evaporator. Among many of factor should concentrate, the heat transfer and pressure drop is most important for performance of PHE. The common assumption in basic design theory that fluid be distributed uniformly at the inlet each fluid side and throughout the core. However, in practice, flow maldistribution is more common and is significantly reduce the desired heat exchanger performance. Nowadays PHE widely use in different industries such as chemical, food process and refrigeration due to the efficient heat transfer performance and the extremely compact design and efficient use of the construction material. In present work PHE will applied in fresh water generator system which installed in ship to convert the seawater to fresh water using the heat from the engines. This paper serves as starting for further research. First provide an overview of PHE cover basic of theory especially focus on pressure drop and flow distribution and second conduct a numerical approach for flow distribution in the channel. The simulation results indicate that pressure and velocity varied sharply around port due to changing of flow area. However at other area the distribution of pressure and velocity is near uniform condition. In other way can found out the tendency that the flow from port to channel then distribute two streams mainly result into there are few fraction at center of channel. Although, it is very difficult to obtain experimental result for comparison with the simulation result but extend detailed comparison with the original experiment and analysis data should carried out within the near future in order to test and further improve the performance of system. That can contribute to the propagate application of plate heat exchanger and it can be practice effective utilization of energy that conserve limited energy.

11:00 ~ 12:20 (Room105)

Free Surface Flows (I)

Session Chair : Prof. D. Wan, Shanghai Jiao Tong Univ/China

M-1E-1. SUSPENDED SEDIMENT TRANSPORT IN 90° OPEN CHANNEL CONFLUENCE

K. DISSANAYAKE, *University of Wollongong, Australia ,* M. SIVAKUMAR, *University of Wollongong, Australia,* A. GODBOLE, *University of Wollongong, Australia*, I. GRASEVSKI, *University of Wollongong, Australia,* Flow dynamics in and around open channel confluences are complex and the presence of sediment will further add to this complexity. Immediately downstream of the junction, the flow develops a zone of separation on the inner wall, with accompanying secondary re-circulation patterns. The structure of this complex flow is a function of several parameters (e.g. flow rates, angle of confluence, sediment concentration) and has a major influence particularly on bed scouring and bank erosion. This makes detailed experimental investigation of such flows very challenging. For investigating these phenomena, experiments were performed in an equal-width, equal-depth, 90° flat bed open channel junction. The downstream tail water velocity and water depth were kept constant, keeping the Froude number closer to 0.37. The sediment (Corvic vinyl) was introduced uniformly to the branch channel and then captured at the downstream end of the main channel. The accumulated sediment was removed from the capture box regularly to facilitate free flow though the fine-grade net. The turbidity level, an indicator of sediment concentration, was estimated using a custom-made optical probe. Higher sediment concentrations were observed adjacent to the inner wall immediately downstream of the junction, indicating particle deposition in the low-velocity separation region. It was observed that with increasing source sediment concentration from the branch channel, the turbidity downstream of the confluence increased while covering a larger area across the width of the main channel. The shape factor, defined as the ratio of separation zone width to separation zone length, was found to vary between 0.12 and 0.15 and has the same order of magnitude as that observed for clean water confluence flow obtained by previous researchers. This experimental study provides valuable information on sediment behavior at channel junctions.

M-1E-2. NON-RADIAL CREEPING FLOW OF POLYMER MELTS THROUGH TAPERED SLIT DIES: AN EXACT SOLUTION

K. SADEGHY, *University of Tehran, Iran*, M. MIRZADEH, *University of Tehran, Iran*, A. PAHLAVAN, *University of Tehran, Iran,* V. ALIAKBAR *University of Tehran, Iran,* S. SADEGHI, *University of Tehran, Iran,* In the present work, it has been shown that even at vanishingly small Reynolds numbers, the assumption of the flow being purely radial might easily be violated in a tapered slit die when dealing with polymeric liquids (polymer melts and solutions). To show this, a series-solution will be attempted to convert the governing PDEs into a set of coupled ODEs assuming that the flow is laminar, two-dimensional, isothermal, and more importantly inertia-less. Two different constitutive equations will be used for the analysis: i) the Giesekus model, and ii) the Phan-Thien-Tanner (PTT) model. Analytical non-radial solutions have been obtained for both fluid models under creeping flow conditions. The analytical solutions so obtained enabled to find the streamline pattern and velocity fields for the fluids of interest. It is shown that for both fluid models, the radial flow assumption is severely violated, particularly near the apex, even at vanishingly small Reynolds numbers. Results obtained in this work suggest that the extensional behavior of a fluid might have a strong influence on the size and intensity of the secondary flows formed near the die exit.

M-1E-3. NUMERICAL SIMULATION OF INTERNAL GRAVITY WAVES GENERATED BY BUOYANCY FORCING IN A CONFINED STRATIFIED REGION

A. A. BIDOKHTI, *Department of Space Physics, Institute of Geophysics,* *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*