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,*