calculated using second order accurate central difference formulae. We employed IB (immersed boundary) technique for the implementation of boundary conditions and semi-implicit fractional-step method for solving the Stokes equations. The flow field and the ionic concentration distributions obtained shows that the electoosmotic effect is predominant in the thin region around the electrode. The initial flow field observed under the application of DC field disappears with the time and under steady state the electrolyte comes to rest. Under AC field the heterocharge layer around the electrodes changes with the frequency of the field. A continuous fluid flow around the electrodes is observed, which can enhance the mixing in the domain. The AC flow field is frequency dependent, a strong flow field is observed in the frequency range 400-600Hz.

11:00 ~ 12:20 (Room103)

Turbulent Boundary Layers

Session Chair : Prof. J. Dey, IIS/India

M-1C-1. ANALYSIS AND PREDICTION OF THE TURBULENT CHARACTERISTICS OF NEAR-SURFACE UNSTEADY WINDS

Jinghong ZHANG, *Key Laboratory of Mechanics on Western Disaster and Environment, Lanzhou University, China,* Xiaojing ZHENG, *Key Laboratory of Mechanics on Western Disaster and Environment, Lanzhou University, China,* Sand saltation movement driven by wind and aeolian sand flow are directly correlated with the characteristics of air flow near earth surface, especially the wind field about 10 cm above it. Wind flow in near-surface atmospheric boundary layer shows evident turbulent characters, and both the lateral wind intensity and its distribution in height are varying with time. In order to make the theoretical prediction of the sediment flux more natural, we carried on experiments performed in bare ground and dune crest at the edge of Badain Jaran desert and Tengger Desert, in China, in which continuous, synchronous measurements of turbulent velocity fluctuations, wind direction, and sediment transport intensity at different height were made. Statistical analysis of the experimental data show that the probability density of the lateral wind gusts approach normal distribution at all heights; turbulence intensity, skewness and kurtosis of the gusty winds all decrease with the increase in height; and sediment transport intensity shows a property of strong unsteadiness and intermittency. Analysis of the fluctuation structure of the wind and the sediment transport intensity was made with VITA method, we found the correspondence were poor, then after smoothing the experimental data, calculation of the correlation coefficient of the localized variance of wind velocity and sediment transport intensity was done, a highest value was found when 3 minutes moving average interval was applied, which reveals a new time scale in the wind erosion process. Finally, we proposed a wind prediction model which can predict wind speed variations of 1Hz at any height below two meters, a good agreement was found when comparing the predicted results and the experimental data.

M-1C-2. RECONSIDERATION OF KARMAN-SHOENHERR SKIN FRICTION FRMULA IN HIGH-RE-NUMBER TURBULENT BOUNDARY LAYER

Hiroki IMANISHI, Kiyoto MORI, Tuji YOSHIYUKI, *Nagoya University*, *Japan,* Tomohiro HATTORI, Masaharu MATSUBARA, *Shinshu University*, *Japan*, Sinsuke MOCHIZUKI, *Ymaguchi University*, *Japan*, Masaru INADA, Tadashi KASHIWAGI, *Kyushu University*, *Japan,* The total and local skin friction of a flat plate is directly measured by using a towing tank up to Reynolds number $Re_L=10^7$ (or $R_\theta=10^4$). Schoenherr (1932) suggested an empirical formula. It is the so-called Karman-Schoenherr formula for total and local frictional resistance. And it is widespread in the shipbuilding research filed and has been highly reliability so far. However, Osaka et al. (1996) showed that the local skin friction, which was measured by the floating element technique, became smaller than the value of Karman-Schoenherr formula. We assume that this discrepancy is due to the Schoenherr's experimental technique. He has ignored the wave-making drag driving from the test plate, the form drag of the plate. In this study, we try to improve Schoenherr's experimental technique, and evaluate the skin-friction coefficient by using the towing tank. Our test plates L=3.30~8.15m in length are towed in still water, balancing the vertical weight by small floats, the draft was varied, where draft is defined as the distance from the bottom of the plate to the surface and the drag force is measured by a highly efficient load cell. We used a tripping wire to promote the laminar-to-turbulent transition of the boundary layer. The experiments were carried out in the towing tank at the Deep Ocean Laboratory of Research Institute for Kyushu University. We have developed the new technique to correct wave-making resistance. The measured total drag is converted into local drag; it is found that the local frictional resistance is 6% smaller than that given by the Karman-Schoenherr formula. This is mainly because he did not correct the additional forces which overestimate the plate resistance. We present the simple correction technique to remove those additional forces, and the corrected local skin friction resistance becomes consistent with that measured by the floating element method.

M-1C-3. TURBULENT HYDRAULIC JUMP OVER A ROUGH BED RECTANGULAR CHANNEL

Noor AFZAL, *Faculty of Engineering, Aligarh Muslim University, India*, A. BUSHRA, *Department of Civil Engineering, University of Nibraska, USA,* The information concerning the effects of boundary roughness on the hydraulic jump is incomplete (Carollo, Ferro and Pampalone 2007, J. Hydraulic Eng. 133(9), pp. 989−999). In the present paper the axial flow structure of turbulent hydraulic jump has been proposed by depth averaged Reynolds mean momentum equations over a rough bed rectangular channel. The averaged normal Reynolds stress closure model of constant eddy viscosity in terms of depth averaged axial velocity in axial distance is proposed. The closed form solution for sequent depth ratio, jump and roller length have been obtained. The sequent depth ratio depends on bed roughness and upstream Froude number. The length jump and roller length as function of sequent depth, are universal relations, which are explicitly independent of bed roughness friction factor. An effective upstream Froude number is also defined where the sequent depth ratio and other hydraulic jump characteristics can be directly deduced from classical hydraulic jump theory, provided the upstream Froude number is replaced by effective upstream Froude number.

M-1C-4. STUDY ON SECONDARY INSTABILITY OF A PLANAR SUPERSONIC MIXING LAYER AT MC=0.5 IN DIRECT NUMERICAL SIMULATION

Faming GUAN, Qing SHEN, *China Academy of Aerospace Aerodynamics, Beijing, China,* There are many approaches to transition in the compressible shear layers, such as secondary instability, bypass and three waves resonant. The secondary instability is one important approach that natural transition. In incompressible shear layer, many works focused on the secondary instability researches. A supersonic planar free shear layer at Mc=0.5 is studied in DNS (direct numerical simulation) methods in present study. The Navier-Stokes equations in perturbation form are solved with a finite difference method of the third order accuracy. The secondary instability is found in the impressible shear layer. Based on the secondary instability, the three-dimensional disturbance wave instability is studied. The developments of the hairpin vortex are simulated. The legs and the heads of the hairpin vortex are bent and drawn in the shear layer when the secondary instability develops and the three-dimensional perturbation waves grow up. In the end of the development of secondary instability, the hairpin vortexes are broken and the frequency splitting happened in the shear layer.

11:00 ~ 12:20 (Room104)

Industrial Applications and Material Processing Flows (I)

Session Chair : Prof. J. Sung, SNUT/Korea

M-1D-1. FLOW DISTRIBUTIONS IN CENTRIFUGAL IMPELLER DEVELOPED FOR AIR-WATER TWO-PHASE FLOW OPERATION

Naoki MATSUSHITA, Tomomichi HASUI, Akinori FURUKAWA, Satoshi WATANABE, Kusuo OKUMA, *Department of Mechanical Eng. Science, Kyushu Univ., Japan,* Centrifugal pumps are utilized in various industrial fields due to its simple structure and easy maintenance, and there is a strong demand to develop a usable one even in the high inlet void fraction under the air-water two-phase flow condition. However conventional pump does not achieve this demand as an impossibility of a pumping-up appears at an inlet void fraction less than about 10%. Authors have investigated air-water two-phase flow performances and clarified several powerful methods to obtain good two-phase flow performances. As the result, a multi-bladed impeller with thin blades, higher outlet blade angle, tandem arrangement of double rotating circular cascades and an installation of diffuser cascade downstream of impeller outlet has been proposed. However, the installation of diffuser cascade downstream of impeller causes the increase of shaft power under the case of water single-phase flow and air-water two-phase flow. It is considered that the increase of circumferential absolute flow velocity due to the installation of diffuser cascade causes the increase of shaft power. Then the flow distribution in the impeller was measured by LDV, and we examined the relation between the increase of shaft power and flow distribution at the impeller outlet. By comparing the results by LDV measurement between the flow distribution in the impeller with diffuser

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,*