H. LIU, *SJTU*, *China*, X. T. DU, *SJTU*, *China*, K. GONG, *SJTU*, *China*, W. WU, *SJTU*, *China,* Landslide-generated water waves have always been of great interest to hydraulic and coastal engineers because the large deformation and breaking of the free surface bring great challenges for theoretical and numerical studies. Smoothed particle hydrodynamics (SPH) is a mesh-free method that offers substantial potential in many classes of problems especially those characterized by large deformations. The two-dimensional underwater landslide is studied numerically by using the Smoothed Particle Hydrodynamics (SPH) method. The experimental data of two dimensional tsunami waves induced by landslides in a flume is used to validate the numerical model.It demonstrated that SPH method can deal with the hydrodynamic flows generated by landslides in reservoir or coastal waters, and the SPH method has the superiority in simulating the phenomena of large deformation of the free surface. The two-dimensional underwater landslide is studied numerically by using the Smoothed Particle Hydrodynamics (SPH) method. Numerical examples show the computed results are very close agreement with the experimental ones. The second plunging appears during the sliding of the box. Two vortexes with different directions were observed above the box after it slide on the bottom of the numerical flume, and the vortex will maintain for a long time even when the water surface become still. In order to study the effects of the sliding speed of the block on the hydrodynamic characteristics, three cases of sliding speed at three values are computed numerically using the SPH model. It turns out that the patterns of the free surface and vortex are different, particularly at the phenomena of reversed plunging.

M-1A-4. THE INFLUENCE OF THE ORIENTATION STRESS TENSOR ON THE BLOOD FLOW IN A VESSEL

G. ZAMAN, *Department of Mathematics, Pusan National University, Korea*, Y. H. KANG, *Department of Mathematics, University of Ulsan, Korea*, I. H. JUNG, *Department of Mathematics, Pusan National University, Korea*, The objective of this paper is to present a short overview of some macroscopic constitutive models that can mathematically characterize the rheology of blood and describe its properties. The three-dimensional Oldroyd-B model coupled with the momentum equation and with the total stress tensor which consists of the isotropic pressure stress tensor, the shear stress tensor and the orientation stress tensor will be presented. Some numerical simulations obtained in geometrically reconstructed vessel will be also presented to illustrate the hemodynamic behavior using non-Newtonian inelastic models under a given set of physiological flow conditions.

11:00 ~ 12:20 (Room102)

Microfluidics (I)

Session Chair : Prof. S. Song, Hanyang Univ/Korea

M-1B-1. A HYBRID MULTISCALE SIMULATION OF MICRO AND NANO-FLUIDICS WITH SURFACE ROUGHNESS

Ping ZHANG, *Laboratory for Nonlinear Mechanics, Institute of Mechanics, Chinese Academy of Sciences, China,* A modified Usher algorithm is developed for hybrid multi-scale simulation of micro- and nano-fluidics with surface roughness. In the hybrid simulation, molecular dynamics (MD) is used in one region where the continuum assumption breaks down, and the Navier-Stokes (NS) equations are used in another region where the continuum assumption holds. A constrained particle dynamics with the dynamic coupling model is introduced to couple the MD and NS equations in the overlap part of those two regions for the mass and momentum continuum. In dynamic coupling model, coupling parameter is flexible and positive as it is dynamically determined by the current states and self-adjusted as computation progresses. When the mass flux obtained from the NS equations is negative, particles nearest to the top boundary should be removed from MD area, when mass flux is positive, additional particles have to be introduced into the MD region to maintain the mass continuum. The critical issue is how to find the locations for the additional particles. In the past, new particles are often placed randomly on the MD region boundary, which causes program stoppage, because some particles are expelled far from the computational region because new particles are too close to the old particles. And solutions to this problem such as long time relaxation are always time-consuming or will cause unsteady temperature fluctuation. The Usher algorithm was developed to search the locations of the additional particles in order to avoid energy to blow up. However, the algorithm is slowly convergent in hybrid method because of the relatively smaller area. In this study, we modify the Usher algorithm to improve its search process. The modified Usher algorithm is used to simulate the Couette flows with surface roughness. The results obtained are found to be in good agreements with the ones obtained from the full MD simulations.

M-1B-2. DROP OSCILLATIONS EXCITED BY POINT FORCE IN AC EWOD

J. M. OH, S. H. KO, K. H. KANG, *Pohang University of Science and Technology, Korea,* The wetting tension is changed by the Maxwell stress concentrated at the three-phase contact line in EWOD (Electrowetting on dielectrics). AC EWOD is preferable in various applications such as LOC (Lab-on-a-chip), liquid lens, and reflective display because it has some advantages such as the delay of the saturation and the decrease of the hysteresis of the contact angle. When the contact angle is changed periodically in AC field, the drop oscillation can be induced. Recently, a few researchers have reported the oscillation of a sessile drop in AC EWOD, and some of its consequences. However, no theoretical analysis of the problem has been attempted yet. In the present paper, we observe the frequency and amplitude dependences of drop oscillations in experiments and propose a theoretical model to analyze the oscillation by applying the conventional method to analyze the drop oscillation. A conventional experimental setup is prepared for observing EWOD. In experiments, the high speed camera is used to observe the instantaneous images of the drop oscillation. It is observed that the drop oscillation becomes resonant at specific frequency and that the dominant shape mode is different at each resonant frequency. It is also observed that the maximum amplitude of the oscillation at resonance decreases with respect to frequency. In theoretical analysis, the conventional domain perturbation method is used to derive the shape mode equations under the assumptions of weak viscous flow and small deformation. The Maxwell stress is exerted on the three-phase contact line of the droplet like a point force. The electrical force is simplified by using a delta function, and is decomposed into the driving forces of each shape mode. In the theoretical analysis, it is observed that the amplitude of each shape mode is maximized at the resonance frequency and that the maximum amplitude decreases with the mode number. The theoretical results on the shape and the frequency responses are compared with experiments, which show a qualitative agreement.

M-1B-3. ELECTROHYDRODYNAMIC FLOWS IN A DIELECTRIC LIQUID

H. J. PARK, J. M. OH, K. H. KANG , *POSTECH, Korea,* It is of great interest to understand the characteristics of colloidal particles under electric field in the dynamics of electrorheological fluid, electrophoretic deposition, and electrophoretic display. The analysis for the dynamics of colloidal particles has been mainly performed in the case of aqueous electrolyte solution. However, if dielectric liquids (DLs) are considered, there is an advantage of little electrolysis. This means that electric field strength can be raised much highly (~10^6V/m) to enhance a dynamic response of colloidal particles. Until now, the dynamics of colloidal particles is poorly understood, in a DL under electric field. In this work, we found a new EHD phenomenon occurring around the colloidal particles. We carried out experiments for a circular-cylindrical rod located in the middle of the electrodes and analyzed the EHD flow around the rod theoretically and numerically. We observed a symmetric flow pattern together with four vortices around the rod. We suggest that the EHD flow will be generated by the gradient of electrical conductivity which is induced by the field-dependent dissociation of impurities in the DL. Experimental result shows good agreement with our numerical results. The numerical solution is verified with analytical solution which is obtained by a perturbation analysis. According to our analytical result, the sign of induced free charges due to the conductivity gradient are negative at the top region and positive at the bottom region of the rod. The corresponding fluid pattern is found to be driven from the ‘equator’ of the rod towards the ‘poles’. The greatest magnitude of velocity is located in the vicinity of the ‘poles’ of the rod where the Coulombic body force is the greatest. The EHD flow is distinct from the conventional electrokinetic flow such as the induced-charge electroosmosis (ICEO), judging from the its flow direction and locations of center of vortices.

M-1B-4. NUMERICAL STUDY OF ELECTROOSMOSIS IN A MICRO-CAVITY USING IMMERSED BOUNDARY METHOD

D. V. FERNANDES, *Dong-A University, Korea,* S. KANG, *Dong-A University, Korea,* Y. K. SUH, *Dong-A University, Korea,* The bulk motion of an aqueous solution induced by the application of electric field is studied numerically. The physical model consists of a micro-cavity with two completely polarizable cylindrical electrodes. The electric double layer (EDL) model coupled with Stokes equations governing the electroosmotic flow has been described. The ionic species distribution is predicted by solving the Poisson-Nernst-Planck equations. All the spatial derivatives are

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