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 (~$10^6$V/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