results show the reasonable agreement with the measured data and it was found some room to optimize the design of radiator for reducing cost. In addition, the sensitivity study for optimizing of the transformer was performed about the heat transfer area of side and bottom end wall in the radiator for obtaining compact size and low cost manufactures. Consequently, the bottom cut case is a little better in entire cooling performance than the side cut case if the other conditions are same due to fluid flow distributed equally.

M-3D-2. TURBULENT HEAT TRANSFER AND PRESSURE DROP IN A TUBE WITH CONICAL-RING AND TWISTED-TAPE INSERTS

V. KONGKAITPAIBOON, *MUT, Thailand,* J. CHAROENSUK, *KMITL, Thailand,* K. NANAN, *MUT, Thailand,* P. PROMVONGE, *KMITL, Thailand,* S. EIAMSA-ARD*, MUT, Thailand,* Influences of the converging-diverging conical-ring (CDR type) inserts on heat transfer and isothermal friction characteristics in a heating tube are investigated experimentally. In the experiments, the CDR Turbulators included three diameter ratios (d/D = 0.5, 0.6 and 0.7) and twisted tapes with two twist ratios (y/w= 3.75, and 7.5) are used for generating stronger turbulence intensity and swirl flow in the tube. The experimental results reveal that the mean Nusselt number and isothermal friction factor increase with decreasing the diameter ratio and the twist ratio. For the Reynolds number ranging from 6,000 to 26,000, the increase in heat transfer due to employing the conical-ring at d/D = 0.5 and twisted-tapes is found up to 250% and to 280% over the plain tube for the CDR with y/w= 7.5 and y/w= 3.75 respectively, while the friction factor is around 81 and 118 times. Based on the same pumping power, the thermal enhancement efficiency of the tube fitted with conical-ring and twisted-tape has also been determined.

M-3D-3. ENHANCEMENT OF PEM FUEL CELL PERFORMANCE BY CATHODE FLOW PULSATION

H. S. HAN, *KAIST, Korea*, S. Y. KIM, *KIST, Korea*, J. M. HYUN, *KAIST, Korea*, A proton exchange membrane fuel cell (PEMFC) is expected as one of the most promising candidates for future power source on account of their high power density, quick start-up and easy operation. In the BOP system, the blower is generally used for uniform air supply to the stack. The supplied air is diffused through the gas diffusion layer (GDL). The performance of a fuel cell is strongly affected by the diffusive mass transport in GDL which is proportional to the reactant concentration gradient between a catalyst layer in MEA and a flow channel in bipolar plate. Also, the limiting current density increases with higher reactant concentration gradient. In the research field on conventional fluid dynamics, the enhancement of heat and mass transport by pulsating flow has been reported. The fluid mixing and heat transfer are enhanced by the periodical convective fluid motion induced by pulsating flow. Thus, the pulsating air supply to the cathode inlet may be considered for higher concentration of oxygen in the cathode flow channel. In the present study, the effect of pulsating cathode flow on the overall performance of a 10-cell PEMFC is investigated. The polarization curve and corresponding power curve are experimentally obtained to identify the effect of pulsating frequency, amplitude and flow rate on the overall performance. The polarization and power curves show that the performance of a 10-cell PEMFC is substantially increased by pulsating cathode flow which enhances the mass transport of reactant in the cathode channels. The increased power output and limiting current density are measured at higher pulsating amplitude. On the other hand, the polarization curve and corresponding power curve is hardly ever changed with the pulsating frequency. The maximum power output increases by 38%, 13% and 5% when the cathode flow rate is 10lpm, 20lpm and 30lpm, respectively. Enhancement of the overall performance is more pronounced at lower flow rate region.

M-3D-4. ACTIVE CONTROL OF TWO STAGE IMPELLER SPEEDS TO SUPPRESS CAVITATION

Kotaro KADO, Kengo SAKAMOTO and Toshiaki KANEMOTO, *Kyushu Institute of Technology*, *Japan,* The cavitation, which is affected by the impeller speed and the suction head, causes the deterioration of the pump performances, the noise and vibration of the pumping system, the erosion of the impeller and so on. Then, the inducer has been installed in front of the main impeller to suppress effectively the cavitation, and the desirable profiles have been proposed. The conventional type inducer attached to the main impeller, however, has a limit in improving the suction performances because the rotational speed of the inducer depends directly on the main impeller speed. Besides, the impeller equipped with the inducer is not suitable for long and/or recycling usages because of the unacceptable erosion of the inducer blade surfaces in the cavity flow. To overcome these week points, the authors have separated the inducer from the main impeller driving system, where the inducer and the main impeller are called hereafter the front and the rear impellers. Both rotational speeds are controlled independently and actively in response to the suction head and the pumping discharge so as to suppress simultaneously the cavitation not only in the rear but also in the front impellers. The performances of the pump, in which the front impeller rotates in the same direction of the rear impeller, were compared with those of the pump, in which the front impeller counter-rotates against the rear impeller. Besides, in order to suppress the cavitation, precisely so that the required NPSH (Net Positive Suction Head) Hre_F of the front impeller coincides with Hre_P of the rear impeller. The required NPSH of both impellers, Hre_F=Hre_P, are markedly low as compared with Hre_M of the commercial pump and the suction performances can be improved successfully. And the front impeller which is counter-rotating against the rear impeller plays better suction performances.

16:30 ~ 17:50 (Room105)

Free Surface Flows (III)

Session Chair : Prof. H. Liu, Shanghai Jiao Tong Univ/China

M-3E-1. EXPERIMENTAL INVESTIGATION ON HYDRAULIC CHARACTERISTICS OF SLUICE CAISSON FOR TIDAL POWER PLANT

D. S. LEE, *Korea Ocean Research & Development Institute, Korea*, S. -H. OH, *Korea Ocean Research & Development Institute, Korea*, J. -H. YI, *Korea Ocean Research & Development Institute, Korea,* H. -S. CHO, *Hyein E & C, Korea,* The basic elements of a tidal power plant, which converts ocean tidal energy into electronic power, are caissons for housing sluices, turbines, and ship locks and barrages that enclose a basin where it is not sealed by caissons. The sluices are opened to allow seawater to flow into the basin by passing through the sluices during the high tide period and then are closed until the basin is emptied after power generation. Hence, the sluice caissons need to be designed for inflowing as many water volumes as possible to maximize the efficiency of power generation. In this study, we carried out hydraulic experiments in an open channel flume and investigated the shape of sluice caisson that is associated with the largest volume of water inflow through the sluice caisson. The experiments were carried out in an open channel flume of 22 m long, 1 m high, and 0.6 m wide. Totally, 15 different caisson models were manufactured by acryl and subjected to the experimental conditions of a variety of local water depth, tidal range, and the seafloor shape around the sluice caisson. The water level in front of and behind of each sluice caisson model and the total water discharge flowing through the sluice was measured precisely with a great care. By analyzing the whole experimental data, it was concluded that the water discharge generally increased by increasing the width of the throat section if the side shape of the sluice was the same. In addition, the water discharge became incremented if the bottom height of the throat section was increased to approximately 30 % of the throat section height. With regard to the length of the throat section, it was advantageous to reduce the length as short as possible, only considering space for the gate structure which is needed for opening and closing of the whole sluice caisson.

M-3E-2. COMPUTATION AND EXPERIMENTS A SLOSHING IN EQUILATERAL-POLYGONAL-SECTION CONTAINERS

Hirochika TANIGAWA, *Department of Mechanical Engineering, Maizuru National College of Technology, Japan*, Masanao GOMON, Tohru NAKASHIMA, Jiro FUNAKI, Katsuya HIRATA, *Department of Mechanical Engineering, Doshisha University, Japan,* When we design various structures with liquid inside, we primarily have to consider the resonance phenomena. Thus, many eigen frequencies f_{mn} of the sloshing is one of key factors. The sloshing is classified into two, namely, horizontal and vertical ones. The former, appears when the periodic force in the horizontal direction is added at an excited frequency $f_0 \approx f_{mn}$. The latter, sometimes referred to as Faraday resonance, appears when the periodic force in the vertical direction is added at $f_0 \approx 2f_{mn}$. In both sloshing, just tiny force can induce the standing wave with a very large amplitude. In the present study, because we have an approach to generalise the sloshing in various shaped containers, we consider the vertical sloshing. In general, the horizontal sloshing is likely to affected by the forcing direction. This study reports the vertical sloshing, that is, the liquid surface motion in container oscillating in the vertical direction, concerning various equilateral-polygonal-section containers: namely, octagonal, heptagonal, hexagonal, pentagonal, square and trianglar containers together with a circular container, in order to generalise their sloshing modes. As a result, the authors classify the sloshing modes based on the circular-container sloshing modes. The stability diagrams for all the polygonal-section containers are

investigated by both experiments and computations. The present computation is based on the discrete singularity method. Furthermore, it is found that the equivalent diameter d_{e1} based on the hydraulic mean depth is the most adequate as a characteristic length scale to classify all the sloshing modes. The authors show an unified formula to predict the eigen frequencies, using the proposed modal classification and d_{e1}.

M-3E-3. ARRESTED TRACER MODEL FOR LONGITUDINAL DISPERSION IN RIVERS

Anton PURNAMA, *Sultan Qaboos University, Oman*, H. H. Al BARWANI, *Sultan Qaboos University, Oman,* A striking feature common to all measured data on longitudinal dispersion in rivers is the existence of a persistent skewness and, in particular, the collected tracer concentration distribution is characterized by an abrupt leading edge and an extended long tail. Another feature is that only a small fraction of the amount of tracer introduced into the river is frequently recovered at the most distance downstream. It is generally accepted that the long tail in the concentration distribution is caused by the tracer arrests by the stagnant zones found at the stream channel, such as sloughed banks and side channels, or behind protruding logs or boulders at the streambed. If more reliable predictive models are to be developed, it would seem important to compare predicted and observed values of the skewness. In the popular dead zone model, the tracer arrests and mixes are formulated using a simple mass exchange mechanism, and by comparing with the results obtained from the field tracer measurements, it is found that it fails to account for the persistence of skewness in observed data. Stream water movement into and out of the hyporheic zone, such as the saturated sediment beneath the stream channel, suggests that the arrested tracer might be transported along complicated pathways before eventually finding its way out, or perhaps not at all. In the arrested tracer model, we assume that the arrested tracer is mixed by a diffusion process in the semi-infinite stagnant zone, and therefore, the tracer spent a long period of time in the stagnant zone. Unlike the dead zone model, the arrested tracer model prediction is not characterized by decreasing the value of skewness. At large times, the model predicts a constant value of skewness, in agreement with the observed data collected.

M-3E-4. COMPUTATIONAL MODELLING OF THE IMPACT OF 2004 TSUNAMI ON THE CITY OF HAMBANTOTA IN SRI LANKA

J. J. WIJETUNGE, *Department of Civil Engineering, University of Peradeniya, Sri Lanka,* On 26th December 2004, coastal belts of Sri Lanka as well as several other countries bordering the Indian Ocean suffered enormous loss of life and damage to property owing to the tsunami unleashed by the third largest earthquake ever recorded. In order to mitigate potential loss of lives from a similar event in the future we need to provide advance warning of an approaching tsunami and then quickly evacuate vulnerable coastal communities to safer areas. Clearly, such evacuation planning requires prior information about vulnerable localities as well as areas that are deemed safe. The information necessary for this purpose is usually obtained through the development of tsunami hazard zonation maps which provide a graphical presentation of the spatial variation of the intensity of the probable depth of inundation and flow velocity across the areas of interest. Accordingly, the present paper outlines the numerical modelling of tsunami propagation and inundation carried out by employing non-linear shallow water equations to develop a high-resolution tsunami hazard map, as a case study, for the city of Hambantota on the south coast of Sri Lanka, which was devastated by the 2004 tsunami. The results give the spatial distribution of the maximum values of the depth of inundation as well as the flow velocities due to an event similar to the 2004 tsunami, which may be considered as a worst-case scenario in the absence of detailed probabilistic assessments of the tsunami threat for Sri Lanka. The model simulations confirm that the sand dunes, where present with sufficient elevation, have helped protect the settlements in their shadow from direct tsunami attack whilst comparatively vast extents of the salterns have acted as sinks to absorb and spread the flood water. The computed tsunami arrival times for the shoreline of Hambantota are also compared with eyewitness accounts.

16:30 ~ 17:50 (Room106)

Multiphase and Particle-Laden Flows (III)

Session Chair : Prof. N. Huang, Lanzhou Univ/China

M-3F-1. NUMERCAL SOLUTION OF THE CAVITATION OVER AXISYMMETRIC BODIES USING THE BOUNDARY ELEMENT METHOD BASED ON POTENTIAL

I. RASHIDI, M. PASANDIDE, N. GHAFOORIANFAR, M. MANSOUR, *Ferdowsi University of Mashhad, Iran,* Cavitation is recognized as an inadvisable problem in most phenomena, but in some circumstances, cavitation is remarked as a beneficial problem. The most important example is the submerged projectiles, in which cavitation is desired because of intense decrease in drag force. The dimensionless parameter which is represented for introducing cavitation is the cavitation number (σ). If bodies move with relatively high velocities inside fluids, cavitation starts at a point in which its local pressure reaches fluid vapor pressure. In low velocities or in high cavitation numbers, cavity is closed over the body and is called partial cavitation. With increase in velocity and decrease in cavitation number, cavity grows and covers all the body, which is called supercavitation. In 1993, Fine and Kinnas devised a nonlinear Boundary Element Method (BEM) based on potential elements for solving partial cavitation flow over a hydrofoil. Partial cavitation flow over torpedoes was conducted by Uhlman et al, using BEM method, and source and dipole distribution over body surface and cavity in 2003. Governing equation on the field of the flow is the Laplace equation. In this method cavitation will be modeled, by means of Green's third identity integral. This equation states that the potential flow on any surface can be shown by means of the ring distribution of sources and dipoles. For this purpose, the rings of the sources are distributed on the cavity surface, and also the rings of the dipoles are distributed on the body and the cavity surface. Applying Bernoulli equation, the relation between the total velocity on the cavity surface, and the cavitation number can be obtained which is called the dynamically boundary condition. The kinematic boundary condition states that the flow does not have any vertical component on the body and the cavity surfaces. In boundary element method (BEM) based on potential, the body and the cavity surfaces are respectively estimated by N_b and N_c number of the elements, which totally form N elements on the aforementioned surfaces. By discretization the governing equation and applying it on the surfaces of the body and the cavity, N number of the algebraic equation is obtained. The unknowns include: N_b number of dipole strengths on the body surface, N_c number of source strengths on the cavity surface, and a cavitation number. Therefore, the numbers of the unknowns are N+1, which is one more than the number of the equations. In order to resolve this problem and also solving the system of equations, an auxiliary equation is needed. To obtain this equation, the definition which states that the algebraic sum of the sources powers on the cavity surface must be equal to zero, is used. . The high velocity and also proper accuracy in calculating the geometry of the cavity are considerable advantages of this method.

M-3F-2. SOLID-LIQUID 2 PHASE HELICAL FLOW THROUGH A SLIM HOLE ANNULUS WITH ROTATING INNER CYLINDER

S. M. HAN, *Sungkyunkwan University, Korea,* N. S. WOO, *Sungkyunkwan University, Korea*, Y. K. HWANG, *Sungkyunkwan University, Korea*, Y. J. KIM, *KIGAM, Korea,* An experimental and numerical investigation was carried out to study solid-liquid mixture upward hydraulic transport of solid particles in a vertical and inclined annulus with rotating inner cylinder. Lift forces acting on a fluidized particle plays a central role in many importance applications, such as the removal of drill cuttings in horizontal drill holes, sand transport in fractured reservoirs, sediment transport and cleaning of particles from surfaces, etc. In this study a clear acrylic pipe was used in order to observe the movement of solid particles. Annular velocities varied from 0.4 to 1.2 m/s. Effect of annulus inclination and drill pipe rotation on the carrying capacity of drilling fluid, particle rising velocity, and pressure drop in the slim hole annulus have been measured for fully developed flows of water and of aqueous solutions. For higher particle volume concentration, the hydraulic pressure drop of mixture flow increases due to the friction between the wall and solids or among solids.

M-3F-3. OPTICAL MEASUREMENT OF VOID FRACTION AND FLOW PATTERNS OF GAS-LIQUID TWO-PHASE FLOW IN A MICROCHANNEL

H. IDE, R. KIMURA, M. KURAUCHI, *Kagoshima University, Japan,* M. KAWAJI, *University of Toronto, Canada,* An optical measurement system was developed to investigate gas-liquid two-phase flow characteristics in a circular microchannel of 100 μm diameter. By the comparison between optical signals obtained by multiple optical fiber probes and video images, mean void fraction was decided successfully. The time-averaged void fraction could be obtained from the time fraction by the passage of gas and liquid phases. These void data were obtained using a T-junction with the same internal diameter as the microchannel but the lengths of the gas and liquid injection lines between the T-junction and flow control valves were quite different in the present experimental conditions of Case 1 and Case 2. The presence of a large compressible gas volume upstream of the T-junction had a significant effect on the two-phase flow characteristics in the microchannel typified by the void fraction data. The effects of the threshold