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