Universiti Teknologi Malaysia, Skudai, The increasing demand for a more powerful microchip has projected the power dissipation for high performance processor up to 300 Watts in 2018. With the dimensions expected to remain at 310 mm^2, the heat density approaches the limit of conventional cooling methods. Since three decades ago, numerous studies on macro jet impingement cooling have been done. However, such approach is not suitable for microchip cooling due to the high velocity impingement, a large nozzle size, and a wide space between the nozzle and the surface to be cooled. Corrosion of the silicon microchip may result and the large size and noise involved make it inapplicable for most of the applications where microchip cooling is required. Microjet array cooling system can be an excellent heat removal mode for microchip particularly from the hot spots. The light and small microjet array enables it to be directly attached on top of the microchip. Unlike the macro fluid flow study, the flow pattern in micro scale is hard to be investigated through an experimental study. This study looks into the numerical simulation of microchip water-cooling with microjet array system impinging with a nozzle diameter ranging from 40 µm to 76 µm. Simulation results show the ability of a single-jet with 76 µm diameter nozzle in cooling the microchip dissipating 4.3 and 6.7 Watts of heat flux over a 1 cm^2 area. As the volumetric flowrate increases from 2 ml/min to 26 ml/min for a single jet, the impinging velocity also increases. The area covered, however, is small compared to the multiple jet arrays. For the multiple jet array, simulation with 4, 9, and 13 jets show that although each jet efficiency is reduced, the overall performance is capable of achieving very low average surface temperature. The simulation showed that at a fixed flowrate, there is a limit to increasing the nozzle diameter to achieve a high heat transfer coefficient. The effect of a larger impingement area is counter-balanced by the effect of lower impinging velocity. Microjet cooling array shows potential in removing excessive heat dissipated from the microchip particularly since hotspots are present in almost all microchip.

M-3B-4. NUMERICAL STUDY OF 2-D NON-NEWTONIAN FLOW
D. TOGHRAIE, A. R. AZIMIAN, *Department of Mechanical Engineering, IUT, Isfahan, Iran,* In this paper behavior of a non-Newtonian turbulent flow between two Parallel Plates is simulated. For this purpose modified Navier-Stokes equations are solved numerically using a finite volume technique. The Quick method was used to approximate the convective terms and the power law model was used to simulate the non- Newtonian fluids. The hydrodynamic developing flow in the entrance region between these plates is obtained and the velocity profiles, pressure gradients, entrance length and Fanning friction factor for a fully developed flow is presented. Numerical results obtained were compared with experimental data and false agreements were found.

16:30 ~ 17:50 (Room103)

Boundary Layer Instabilities

Session Chair : Prof. M. Asai, Tokyo Metropolitan Univ/Japan

M-3C-1. INFLUENCE OF AN UNFAVOURABLE PRESSURE GRADIENT ON THE BREAKDOWN OF BOUNDARY LAYER STREAKS
V. V. KOZLOV, *Institute of Theoretical and Applied Mechanics SB RAS, Russia*, V. G. CHERNORAY, *Chalmers University of Technology, Sweden*, I. LEE and H. H. CHUN, *Naval Architecture & Ocean Engineering, Pusan National University, Korea,* Experimental studies of nonlinear instabilities of boundary layer streaks are discussed. Extensive measurements visualizing the sinusoidal and varicose instabilities of streaks at nonlinear stages of breakdown process are presented. Specific features of the development of the streamwise streak breakdown are demonstrated, and various scenarios of the origination and development of coherent vortex structures are discussed.

M-3C-2. INCIPIENT SPOT IN A BLASIUS BOUNDARY LAYER: SOME ASPECTS
R. SUR, *Department of Mechanical Engineering, Jadavpur University, Kolkata, India,* A. C. MANDAL and J. DEY, *Department of Aerospace Engineering, Indian Institute of Science, Bangalore, India,* In an incipient spot that propagates creating spanwise streaks, both the streamwise energy growth rate and the collapse of different streamwise wave number spectra at different downstream stations are found to be similar to those reported for boundary layers subjected to high freestream turbulence.

M-3C-3. THE EFFECT OF OUTER DISTURBANCE ON TRANSITION OF A FLAT-PLATE BOUNDARY LAYER
M. SHIGETA, T. OHNO, S. IZAWA, Y. FUKUNISHI, *Department of Mechanical Systems and Design, Tohoku University, Japan,* The effect of the outer disturbance on the transition of a flat-plate boundary layer is investigated by a wind-tunnel experiment. The outer disturbance is introduced from the outside of the boundary layer downstream of the leading edge to avoid the leading edge receptivity. The relation between the characteristics of the outer disturbance and the growth of the velocity fluctuation inside the boundary layer is studied. A flat plate with a leading edge modified to reduce the receptivity is horizontally mounted in the test section of a low-turbulence wind tunnel. The free stream velocity U_∞ is 5.0 m/s and the freestream turbulence is less than 0.25 %. The external disturbance is introduced by a turbulence-generating bar which has small holes opened on one side. From these holes, jets are issued at the mean velocity of 9.8 m/s in the direction parallel to the freestream. The measurements are conducted by a standard single hot-wire probe. When the coflow jets are injected from the turbulence-generating bar, at first, the turbulent region only gradually spreads while traveling downstream. However, at a location, the velocity fluctuation jumps into the near-wall region of the boundary layer creating a new source of disturbed flow at the wall. It is found that particularly the low frequency component of the fluctuation jump, and that jump is done not directly toward the wall but obliquely tilted in the spanwise direction. The localized disturbances at the wall are replaced downstream by the disturbances generated by the streaky structures, which lead to a boundary layer transition.

M-3C-4. HEAT TRANSFER CHARACTERISTICS OF A MINIATURE LOOPED PARALLEL HEAT PIPE
C. M. FEROZ, M. E. HOQUE, M. N. ANDALIB, *BUET, Bangladesh,* Heat pipe or thermosyphon is a device of very high thermal conductance. Among other cooling techniques heat pipe emerged as the most appropriate technology and cost effective thermal design due to its excellent heat transfer capacity, high efficiency and structural simplicity. Heat pipe can, even in its simplest form, provide a unique medium for the study of several aspects of fluid dynamics and heat transfer, and it is growing in significance as a tool for use by the practicing engineer or physicist in applications ranging from heat recovery to precise control of laboratory experiments. Thermal designers have widely accepted the miniature heat pipe for their thermal design solutions and the area of application is increasing day by day. The present experimental work investigates the heat transfer performance of a miniature looped parallel heat pipe[MLPHP] which consists of two single tube heat pipes connected by two U- tubes of same diameter at the top and bottom ends. For this purpose, the copper tube of 5.78 mm ID is used with methanol as the working fluid. Analysis of the experimental data gives that the axial wall temperature of both condenser and evaporator sections increase with increase in heat flux and decrease with the increase in coolant flow rate. The thermal resistance of MLPHP decreases with the increase of both coolant flow rate and thermal load. Overall heat transfer coefficient increases with the increase of both coolant flow rate and heat flux.

16:30 ~ 17:50 (Room104)

Industrial Applications and Material Processing Flows (III)

Session Chair : Prof. S. Fu, Tsinghua Univ/China

M-3D-1. NUMERICAL SIMULATION FOR OPTIMIZING THE OIL COOLED DISTRIBUTION TRANSFORMER
S. W. YANG, W. S. KIM, K. Y. KWEON, H. S. LEE, *Power & Industrial Systems R&D center, Hyosung Corporation, Korea,* This paper describes the numerical simulations in the cooling of the radiator in a distribution transformer. The aim of this work is the cooling optimization of the transformer by CFD simulations. A clear understanding of the cooling pattern in a radiator which is a main heat remover in the power transformer is essential for optimizing the radiator design increasing the thermal efficiency. In this paper we study the heat transfer and fluid flow in a 3-phase 400kVA transformer. The plate radiators of this transformer become wrinkled (corrugated radiator) and there are filled with transformer oil. The oil is circulated due to the natural convection driven by buoyancy effects through radiators so that the ultimate cooling medium is the surrounding air. In the design of transformers, it is of interest to minimize the cost and size of radiators. The obtained results show the temperature and flow distributions and the possibility to optimize the transformer with 3-dimensional CFD models using FLUENT. For designers of transformer, the temperatures of interest are top oil rise, average oil rise, average winding rise and hot spot. There are design limits for the temperature rise such as 50°C for oil and 55°C for winding in this kind of transformer. The obtained

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