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 cm² 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