ratio of retardation time to relaxation time and τ, the ratio of the diffusivities, are greater than unity. In contrast to the single component system, it is found that a small amount of throughflow in either of its direction destabilizes the system. It is also observed that increase in the value of elasticity parameter Γ, and decrease in the values of Λ, τ and solute Rayleigh number R_s, is to hasten the onset of convection.

M-3A-3. VISCOELASTIC EFFECTS OF POLYMER SOLUTIONS ON OIL RECOVERY ENHANCEMENT IN CHEMICAL FLOODING

Z. ZHANG, J. C. LI, J. F. ZHOU, *Institute of Mechanics, CAS, China,* Polymer is a commonly used chemical reagent to Enhance Oil Recovery (EOR) in chemical flooding. The present dominant view on the mechanism of polymer is that the polymer solutions can improve sweep efficiency by enhancing mobility ratio between oleic and aqueous phases. However, when flowing through porous media, polymer solution tends to display viscoelastic effect. So it is necessary to further examine visco-elastic effect of polymer solution in the process of oil exploitation. In this paper, a constitutive equation of the viscoelastic polymer solution is established at first. In order to study the rheological properties of polymer solution, experiments were performed. The result was that the viscosity measured with viscometer is smaller than the viscosity obtained when the polymer solution flows in porous media pores. It is thought that the viscosity difference comes from the elastic effect of polymer solution. Normally polymer molecules exist in solution as loose, roughly spherical coils. When polymer solution passes through tortuous and converging/diverging pore channels, it may stretch or elongate in the direction of flow. At this time polymer solution exhibits an additional viscosity due to such kind of elastic response, namely, the effect of elasticity. So we call the viscosity difference as elastic viscosity. When the polymer solution flows in porous media, elongational flow coexists with shear flow. So the apparent viscosity of a polymer solution consists of shear viscosity, μ_{vis}, and elastic viscosity, μ_{elas}. Then we apply finite difference method to numerically simulate the visco-elastic flow of polymer solutions in porous media. The result shows that the oil recovery of viscoelastic effect is higher than that of viscous effect in the primary stage of displacement process, but the two ultimate recoveries are basically equal. The reason is that due to the viscoelastic effect of polymer solution; more residual oil left in pores during water flooding is hauled out in the process of displacement at the beginning of the displacement. In the chemical flooding with viscous effect, the residual oil saturation can also be reduced if the injected polymer solution is more enough or injection time is long enough. But the production cost will increase when more polymer solution is injected to the reservoir or the exploitation lasts longer. And we must consider the exploitation costs during oil extraction. So the chemical flooding with viscoelastic effect is more practical than that of viscous effect from the point of saving cost. The paper also analyzes the effect of major physical variable on displacement efficiency in details in visco-elastic polymer flooding. On the basis of numerical simulation, we find that relaxation time of polymer accounting for visco-elastic effect plays a decisive role in chemical flooding.

M-3A-4. FRONTOGENESIS IN NONLINEAR STUDY OF STRATIFIED POORLY CONDUCTING FLUID FLOW THROUGH POROUS MEDIA

C. V. VINAY, *J.S.S.Academy of Technical Education, Department of Mathematics, India,* N. RUDRAIAH and G. RANGANNA, *National Research Institute for Applied Mathematics (NRIAM) Bangalore India and UGC-CAS in Fluid Mechanics, Department of Mathematics, Bangalore University, India,* An analysis is presented in this paper to find the exact solution of modified nonlinear Darcy-Lapwood Forchheimer (DLF) equation governing the motion of a heterogeneous poorly conducting fluid saturated porous media in the presence of an electric field and nonuniform vertical density gradient caused by contaminant. These contaminants allow variation in density having fixed charge density (FCD) making the liquid phase poorly conducting and possess significant vapor pressures. If these vapors are denser than air, then eventually sink down to the capillary fringe to contaminate groundwater and also making non potable. We show in this paper the contaminant transport in a porous medium can be resolved using frontogenesis (i.e., increase in horizontal density gradient) developed in a poorly conducting fluid in the presence of self-generating electric field arising from geomagnetic phenomena. For this we use modified Darcy-Forchheimer equation for heterogeneous fluid, together with the Maxwell equations valid for poorly conducting two dimensional fluid flow with electrical conductivity σ of the form $\sigma = \sigma_0\left[1+\alpha_h\left(T-T_0\right)+\alpha_c\left(C-C_0\right)\right]$, where T is the temperature, and C the concentration of mixture α_h and α_c are the volumetric coefficient for σ. The solution of nonlinear momentum equation is determined using the time evaluation of this system. The streamlines and isopyenals are computed for different values of density discontinuity gradients γ_1, γ_2 and the results are represented graphically for different values of time t and for electric number W_1. We found that the streamlines are closer together for $x < 0$ than for $x > 0$ and the density profiles are crowded at the lower region depicting an increase in density gradient and beginning of frontogenesis. The density profiles reveal the curvature near $x = 0$ which develops a circulation in the transverse plane revealing an increase in the magnitude of the density gradient with an increase in time.

16:30 ~ 17:50 (Room102)

Microfluidics (III)

Session Chair : Prof. S. Honami, Tokyo Univ/Japan

M-3B-1. A NOVEL MODEL FOR HEAT CONDUCTIVITY OF NANOFLUIDS

E. SHIRANI, *Isfahan University of Technology, Iran*, S. NABI, *Isfahan University of Technology, Iran*, In this paper, Brownian motion of nanoparticles and clusters and resulted micromixing are combined with the aggregation kinetics of nanoparticles and formation of clusters to capture the effects of added nanoparticles on k_{eff}. Results show that the observed anomalies reported in experimental works can be well described by taking aggregation kinetics into account. The proposed model, attribute the effective thermal conductivity not only to the intrinsic physical properties such as thermal conductivity of the liquid and nanoparticles, viscosity of the liquid, and density of the nanoparticles, as well as temperature and time, but also to physicochemical parameters which affect stability state of nanofluids such as the Hamaker constant, the surface charge, *pH*, and ion concentration. The more nanofluid is stabilized, the more k_{eff} will increase. We have also demonstrated that the thermal conductivity ratio can also increases with particle size depending on the chemistry of the solution. Consequently, an optimized radius in a suspension with certain temperature and *pH* can be achieved. This behavior is not feasible without including the effects of aggregation kinetics combined with Brownian motion and induced micro-convection.

M-3B-2. ON THREE DIMENSIONAL STRUCTURE OF VISCOELASTIC FLUID FLOWS IN A CURVED MICROCHANNEL

F. -C. LI, *Harbin Institute of Technology, China*, H. KINOSHITA, *The University of Tokyo, Japan*, M. OISHI, *The University of Tokyo, Japan*, T. FUJII, *The University of Tokyo, Japan*, M. OSHIMA, *The University of Tokyo, Japan*, Solutions of flexible high-molecular-weight polymers or some kinds of surfactant can be viscoelastic fluids. The elastic stress is induced in such viscoelastic fluids and grow nonlinearly with the flow rate and results in many special flow phenomena, including purely elastic instability in the viscoelastic fluid flow. The elastic flow instability can even result in a special kind of chaotic flow motion, the so-called elastic turbulence, which is a newly discovered flow phenomenon and arises at arbitrary small Reynolds number. In this study, we experimentally investigated the three dimensional irregular flow structures of viscoelastic fluids in a curved microchannel by means of high-speed confocal micro-particle image velocimetry (PIV) as well as visualization technique. The viscoelastic working fluid was aqueous solutions of surfactant, CTAC/NaSal (cetyltrimethyl ammonium chloride/Sodium Salysilate). For comparison, water flow in the same microchannel was also tested. The Reynolds numbers for all the microchannel flows were quite small (for solution flows, the Reynolds numbers were much smaller than 1) and the flow should be definitely laminar for Newtonian fluid. High-speed confocal Micro-PIV, combining with visualization technique, has been used to investigate the flow structures. Figure shows the schematic of the curved microchannel and the micro-PIV measurement location. Figure 2 demonstrates the measured velocity field in a plane at different height from the channel bottom. It was found that the regular laminar flow pattern for low-Reynolds number Newtonian fluid flow in the curved microchannel was strongly deformed in the viscoelastic solution flows, which behaved an apparently three dimensional flow structures. This phenomenon was considered to be induced by the viscoelasticity of the CTAC solution. Further discussions on the viscoelasticity-induced flow behaviors of the solution flows have been made.

M-3B-3. NUMERICAL SIMULATION OF A MICROCHIP COOLING WITH MICROJET ARRAY

Y. E. SOON and M. G. NORMAH, *Faculty of Mechanical Engineering,* *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