tested to simulate linear wave oscillation in *X*, and *X-Y* oscillations. The results show that the scheme is significantly less dissipative especially in *X-Y* oscillation. For simulation along *X* direction only, the scheme shows significantly reduce dissipation when Courant number is less than 0.5. Simulations of more complex wave (two waves of different length) oscillations in *X-Y-T* space were conducted at low resolution. The longer wave is represented by 16 grid points whilst the shorter is represented only by 8 grid points. The results indicate that the scheme is superior when compare to the previous characteristic schemes. After even 20 oscillations of the longer wave (40 oscillations of the shorter wave), the group can still be easily recognized. On the other hand, the shorter wave dies out after 20 oscillations with the original quadratic interpolation scheme. The difference of the numerical wave speed between the longer wave and the shorter one is easily observed. It is concluded that the new interpolation scheme in *X-Y-T* space is convergence at appropriate value of *K* and *f*. It also improves the performance of the characteristic scheme in *X-Y-T* space.

M-2G-3. AN INVESTIGATION AND COMPARISON OF ROE UPWIND METHODS WITH CUSP CENTRAL DIFFERENCE SCHEMES

Mahmood P. FARD, *Ferdowsi University of Mashhad, Iran*, M. SALARI, M. MANSOOR, M. Malek JAFARIAN, *Birjand University, Iran,* In this paper, Euler equations have been solved for the internal compressible flows. In order to have such a solution, at first, the equations were written into the integrated form, and then in CUSP and Scalar schemes, by means of the central difference method, and in Roe scheme, by means of the upwind difference method, they were made into the discritized form. The main problem in the central difference method for solving these equations that occurs as a result of the shock phenomenon is instability and discontinuity of the solution, which is due to the misuse of the information related to the both sides of the shock. In order to solve this problem, we have to apply the artificial dissipations. However, the upwind methods based on the distribution of the flow information along the defined directions in the physical domain; do not have such a problem. In this paper, these schemes will be introduced, and the results of these methods will be compared and then the CUSP scheme will be amended and we will study the performance of central and upwind schemes in solving Euler and Navier-stoks equations. Since, the central difference schemes do not imply enough dissipation, the artificial dissipation, which has an important effect on the accuracy of the solutions, should be added to the equations. The second group is the upwind methods. These methods are based on the distribution of the flow information along the defined directions in the physical domain. So, these methods have a good coincidence with the physics of the flow information all over the field of the fluid flow. After representing Roe and CUSP schemes, we will compare them and their results in solving Euler equations in a nozzle. The results of these methods will be represented for flows with different Mach numbers. The results which have been obtained show that CUSP scheme for low supersonic flows, comparing with the other methods, leads to more accurate results to capture the shock. However, for the higher Mach numbers, it seems it needs some corrections. Thus the modified CUSP scheme which is based on the CUSP itself, reveals more accurate results the high Mach numbers in the vicinity of shock.

M-2G-4. NON-LINEAR MIXED-OBERBECK ELECRTOCONVEC TION IN A POORLY CONDUCTING FLUID THROUGH A VERTICAL CHANNEL

B. S. SHASHIKALA, *Department of Mathematics, Siddaganga Institute of Technology, India*, N. RUDRAIAH, *Centre for Advanced Studies in Fluid Mechanics, Department of Mathematics, Bangalore University, Bangalore,* The study of combined free and forced convection called mixed convection (MC) in a vertical channel has received considerable attention because of its wide range of applications from cooling of electronic devices, gas-cooled nuclear reactors to that of solar energy collectors. In contrast to MC due to variation of density with temperature discussed above there is another type of MC arising in a poorly conducting alloys like nickle-titanium(N_i-T_i), aluminum oxides and so on in a vertical channel due to variation of electrical conductivity, σ ,with temperature in the presence of an electric field, $\vec{E}$,called mixed electroconvection (MEC). The variations of σ with temperature releases the charges forming distribution of charge density, ρ_e. These charges in turn induce the electric field, $\vec{E}_i$ called thermal or induced electric field. In addition there may be an applied electric field, $\vec{E}_a$, due to embedded electrodes of different potentials at the boundaries. The total electric field $\vec{E}(=\vec{E}_i+\vec{E}_a)$ in a poorly conducting fluid produces a current, which acts as sensing. In addition this $\vec{E}$ together with ρ_e produces an electric force $\rho_e\vec{E}$ which acts as actuation. These two properties are the two important properties required a material to be a smart material. Therefore the control of MEC plays a significant role to synthesize smart materials for practical use in science, engineering and technology. Is spite of these importances much attention has not been given to the study of MEC in a vertical channel and is studied in this paper. The nonlinear-coupled momentum and energy equations are solved numerically using finite difference technique and analytically using regular perturbation method with B_r as the perturbation parameter. The analytical and numerical solution are in good agreement. These results in the presence of total electric field, viscous and Joulean dissipations obtained are useful in the effective control of heat transfer in many industrial problems.

16:30 ~ 17:50 (Room101)

Flows in Porous Media (II)

Session Chair : Prof. J. Zhou, CAS/China

M-3A-1. REQUIRED HEAD FOR LOOSE-FINE MATERIAL FLUIDIZER SYSTEM

R. TRIATMADJA, *Gadjah Mada University, Indonesia*, A. THAHA, *Hasanuddin University, Indonesia*, N. YUWONO, *Gadjah Mada University, Indonesia*, NIZAM, *Gadjah Mada University, Indonesia,* Maintenance dredging and channel protections using coastal structures may solve the problem of sedimentation although they may not always be cost efficient. An alternative approach namely fluidization technique may be used to fluidize the sediment and flush the sediment out in to the sea. In this paper, the required head for full fluidization stage is studied using one dimensional, two dimensional and three dimensional fluidizer system models. The one dimensional model is used to study the initial and full fluidization in a tube. Two dimensional models are utilized to study the required head for full fluidization at various condition of sediment height (d), water depth (h), diameter of the perforation holes (D_f), and the distance of the perforation holes (a). Finally three dimensional full scale models are constructed to study the required head for full fluidization. The scale model is designed so that the scale effect of the model is minimized. The study shows that the distance of perforation holes and the diameter of the perforation holes play important roles. The longer is the distance between the perforation holes the less is the sharing of the jets and hence, higher required pressure head. Smaller perforation holes require higher head to maintain effective jets' strength. The required head for full fluidizations h_f/d_b is found to be strongly correlated with $D_f / (a.d_b)^{0.5}$ *a.* The head requirement at full fluidization may also be shown as a function of D_f/a. When plotted together with data from previous results it is apparent that all the data have similar trend where h_f/d_b is decreasing with increasing D_f/a. The theoretical line agrees quite well with the present experiment. Finally it may be concluded that the required head for full fluidization may be represented by the following equation: $\frac{h_f}{d_b} = p\frac{\sqrt{a\,d_b}}{D_f}$, where *p* is approximately 0.23.

M-3A-2. DOUBLE DIFFUSIVE CONVECTION IN A NON-NEWTONIAN FLUID SATURATED POROUS LAYER WITH THROUGHFLOW

S. SURESHKUMAR, *Department of Mathematics, Siddaganga Institute of Technology, India,* I. S. SHIVAKUMARA, *Department of Mathematics, Bangalore University, India,* The effects of quadratic drag and vertical throughflow on the onset of double diffusive convection in a non-Newtonian fluid saturated horizontal porous layer are investigated. The study finds its applications in chemical engineering, geothermal systems, enhanced recovery of petroleum reservoirs, underground spreading of chemical wastes, sea bed hydrodynamics and more importantly in the directional solidification of alloys, where a mushy zone exists which is regarded as a porous layer with double diffusive origin. Many practical problems cited above involve non-Newtonian fluid flow through porous media which is based on a generalized Darcy equation. There exist many different types of non-Newtonian fluids. However, some oil sand contains waxy crude oil at shallow depths of the reservoirs which are considered to be viscoelastic fluids. In such situations, a viscoelastic model of a fluid will be more realistic than inelastic non- Newtonian fluids. Also, many geophysical and technological applications involve non-isothermal flow of fluids through porous media, called throughflow. A modified Forchheimer-extended Darcy model which takes inertia into account and viscoelastic effects is employed to describe the flow in a porous medium. The boundaries are considered to be impermeable and perfect conductors of heat and solute concentration. Conditions for the occurrence of stationary and oscillatory convection are obtained analytically using the Galerkin technique. It is shown that oscillatory convection occurs even if Λ, the

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,*