dune, while the migration speed is always an unsteady quantity controlled by wind speed and diameter of sand particles, which is still not clear. Considering the limitations of existing dune field model, this work reports on a three-dimension model with several modifications involved: (1) Inspired by the field measurements of Finkel (1959) and Sauermann et.al (2000) in Peru and Morocco, we introduce the relationship between dune height and windward slope angle; (2) According to Lancaster's conclusion (1995) that wind speed-up factor changes with windward slope angle and dune height, we put forward a logarithm relation between wind speed-up factor α and the dune height; and (3) We take account of the effect of wind acceleration process on formation and evolution of dune field not only in determining the transport length L_{ij}^t but also the thickness h_{ij}^t of sand slabs. Such modification is proved to be significant in simulating the realistic character of wind-formed features. Moreover, the evolution process of dune field in different desert region is proved to have obvious discrepancy.

W-2E-3. EFFECTS OF AEROSOL SIZE AND DEFORMATION ON CLOUD FORMATION IN THE ATMOSPHERE

N. DEVARAJU and N. RUDRAIAH, *UGC-Centre for Advanced Studies in Fluid Mechanics, Bangalore University, India,* There is a growing concern that human activities may alter the climate by releasing a large amount of soot and other pollutants into the atmosphere in the form of ultra fine dust particles which are suspended in the atmosphere called aerosols. Atmospheric aerosols play the important role in the atmospheric processes of favoring cloud formation and also negatively linked to a number of undesirable phenomena ranging from visibility reduction to adverse effects on human body depending on their size due to coagulation. The coagulation causes aerosol hit each other leading either to stick each other resulting in increase in size and decrease in number or collide each other, leading to the formation of tiny particles resulting in decrease in size and increase in number but in both the cases mass concentration remains the same. Knowing the complexity of aerosols due to coagulation, their favorable or unfavorable effects and the desire to control atmospheric aerosol, the study dispersion of aerosols is crucial. This is done in this paper considering large size aerosols as the mixture of agglomeration and coalescence in the atmosphere and modeled them as fluid saturated sparsely packed porous media. Using this assumption and considering aerosols as deformable the required basic equations are derived incorporating advection and diffusion using mixture theory and Saffman dusty fluid model. The solutions of the basic equations are obtained using regular perturbation technique together with Saffman slip condition on velocity and the permeable condition on concentration. The Taylor dispersion coefficient D_r is obtained and is computed for different values of dimensionless number $R_i (i = 1,2,3)$ and the results are tabulated. From this table we conclude that D_r decreases with an increase in R_4 where R_4 has the dimension reciprocal of Reynolds number.

W-2E-4. A NUMERICAL SIMULATION OF DUST DEVIL AND ELECTRIC FIELD IN IT

N. HUANG, G. W. YUE, X. J. ZHENG, *Key Laboratory of Mechanics on Western Disaster and Environment, Lanzhou University, China,* On a hot and dry day in arid regions, it's common to see swirls of dust race across the landscape. The strong electric field of dust devils may be a possible nuisance or hazard to future human explorers on the surface of planets (Farrell et al., 2004), and therefore it is important to study electric field of field dust devils. Because it is difficult to get the detailed information of the electric field in dust devils through measurement, the simulation becomes an effective way to study the electric field of field dust devils. In this paper, based on the surface energy-balance equation and atmospheric movement equations, and Coulomb's law, the whole process of dust devil development and the electric field in dust devil is numerically simulated, then the simulated results of electric field are discussed and compared with field measurements. It is found that the simulated electric field agrees well with the measured result when the charge-mass ratio of sand grains with diameters of 0.15mm、0.2mm and 0.25mm are taken as $-120\mu C/kg$、$60\mu C/kg$ and $57\mu C/kg$, respectively. The results also show that for electric field in dust devil, it needs about 80s from the moment when some sand particles begin to be lifted off from bed (t=0) to the stage that the value of electric field becomes relatively stable. The absolute value of electric field at a given height always increases as the radius decreases and it will reach a maximum value at the center of dust devil. The absolute value of electric field in dust devil increases first and reaches a maximum at the height of 20m and then decreases with height.

10:40-12:00 (Room106)

Granular Flows

Session Chair : Prof. K. Hirata, Doshisha Univ/Japan

W-2F-1. NONLINEAR STABILITY OF GRANULAR SHEAR FLOW: LANDAU EQUATION AND SHEAR-BANDING

Priyanka SHUKLA*, Engineering Mechanics Unit, Jawaharlal Nehru Centre for Advanced Scientific Research, India,* Meheboob ALAM*, Engineering Mechanics Unit, Jawaharlal Nehru Centre for Advanced Scientific Research, India*, Starting from the continuum equations of rapid granular flows, we derived Landau equation for the plane Couette flow using both the amplitude-expansion method and the center-manifold reduction. Our amplitude `order-parameter' equation describes the onset and the subsequent dynamics of shear-band formation near the critical point. To find the actual behavior of flow due to finite-amplitude disturbances, we need to calculate Landau coefficient which can be expressed in terms of a suitable inner-product of the nonlinear terms and the eigenfunctions of the related adjoint problem. The numerical results on Landau coefficients suggest that there is a sub-critical *finite-amplitude* instability for dilute flows even though the dilute flow is stable according to the linear stability theory. This result is in agreement with previous molecular dynamics simulations of granular Couette flow as well as with the direct solution of nonlinear continuum equations. The scaling of equilibrium amplitudes with different control parameters as well as the effects of mean-flow distortion will be discussed.

W-2F-2. LARGE-SCALE STRUCTURES AND FLUCTUATIONS IN 3D GRANULAR POISEUILLE FLOW

Ashish MALIK and Meheboob ALAM, *Engineering Mechanics Unit, Jawaharlal Nehru Centre for Advanced Scientific Research, India,* Various macro- and micro-structural features of a gravity driven three-dimensional (3D) granular Poiseuille flow are studied in the rapid flow regime using event-driven simulation. A monodisperse system of rough, inelastic hard spheres interacting via hard core potential is considered. The collisions are assumed to be binary and instantaneous in which only momentum is conserved but the energy is non-conserved quantity. The wall-particle interactions are modeled using same collision dynamics as for particle-particle interaction. The structure formation in the form of large-scale density waves takes place under certain conditions. These density waves are affected by various parameters like the volume fraction, the coefficient of restitution and the aspect ratio of the simulation domain. In particular, these structures are driven by inelastic collisions of particles and hence these are dissipation-induced structures.

W-2F-3. INFERENCE OF FACTOR OF TRANSPORTATION EFFICIENCY IMPROVEMENT BY ROTARY FEEDER IN PLUG CONVEYING

K. KOFU, M. OCHI and M. TAKEI, *Department of Mechanical Engineering, College of Science & Technology, Nihon University, Japan,* In plug conveying system, it is desired that the transportation efficiency is improved. Then a rotary feeder is often used these days. However, the reason of the transportation efficiency improvement has not been clarified. Therefore this reason has been investigated experimentally. Two kinds of pipe diameters, i.e., 38 and 50 mm, were used. The pipe line length was about 11.4 m. Four kinds of particle-air mixing devices were used, and feeders were changed to a rotary feeder and a vessel in each experimental condition. As a result, there is no improvement of transportation efficiency by the rotary feeder in all experimental conditions, because there is little difference between the largest particle mass flow rate by the rotary feeder and that by the vessel. However, the deviation of plug length and velocity in a rotary feeder is smaller than that in a vessel under all experimental conditions. This is assumed as the improvement reason. In short, the number of incorporated plugs during transportation in a rotary feeder is smaller than that in a vessel. In this case, plug length are short, large air mass flow rate is not required to transport particles in a rotary feeder and transportation efficiency becomes large. It is thought there is no improvement because the total pipe length is short and combinations of plugs are seldom used in this study. Additionally, it is said that the effect by pipe diameter and volume of particle-air mixing device on transportation efficiency improvement is small. Some experimental conditions show that particle mass flow rate is not dependent on the number of vane rotation of a rotary feeder. This reason is also considered from the result of the particle velocity measured by the high speed camera and PIV in the particle-air mixing device. Particles in the upper part move smoothly in spite of the experimental conditions and kinds of particle-air mixing device, although it is difficult to move lower particles in two kinds of particle-air mixing device. However, all particles flow smoothly in the bend type device which shape is curve in spite of air mass flow rate. In short, the particle-air mixing device affects the particle transportation efficiency when the rotary feeder is used, and a device that does not have a flat base should be used in order to make use of the merits of a rotary feeder.

W-2F-4. MARANGONI MAGNETO ELECTROCONVECTION IN A POORLY CONDUCTING FLUID-SATURATED POROUS LAYER COOLED FROM BELOW IN THE PRESENCE OF THE ELECTRIC AND MAGNETIC FIELDS

N. RUDRAIAH, *UGC-CAS in Fluid Mechanics Department of Mathematics, Bangalore University Bangalore and National Research Institute for Applied Mathematics, India,* C. O. NG, *Department of Mechanical Engineering, the University of Hong Kong, China,* The combined effect of Hartmann, Brinkman and electro boundary layers on the onset of electroconvection driven by the combined electromagnetic force and surface tension gradient, called Marangoni magneto electroconvection (MMEC), in a thin horizontal Poorly electrically conducting fluid saturated sparsely packed porous layer bounded by adiabatic free boundaries is studied analytically using a linear stability analysis. The moment and energy methods combined with single term Galerkin expansion technique are shown to be convenient and useful to determine eigenvalues. The results obtained are compared with those obtained by Rudraiah et.al [1] in the absence of magnetic field. We found that the single-term Galerkin expansion gives reasonable results for small values of combined porous parameter, Hartmann, and electric parameter d0<10 . Also we found that an increase in Hartmann number increases the critical Marangoni number and hence it suppresses MMEC. The results obtained are useful in the manufacture of smart materials of nanostructure free from impurities.

10:40-12:00 (Room107-108)

Computational Fluid Dynamics (VIII)

Session Chair : Dr. C. S. Ahn, ADD/Korea

W-2G-1. FLOW FIELD AND PERFORMANCE IMPROVEMENT OF A CROSS FLOW FAN - EFFECT OF CASING GEOMETRY

M. GOVARDHAN and D. Lakshmana SAMPAT, *Thermal Turbomachines Laboratory, Department of Mechanical Engineering, Indian Institute of Technology Madras, Chennai, India,* This present paper describes three dimensional computational analysis of complex internal flows in a cross flow fan, (CFF) with a special emphasis on the performance improvement by varying casing geometry. Simulation of flow through the CFF is done using CFD software package, CFX-5. Geometry modeling and grid generation are done in CFX-Build and the necessary boundary conditions are incorporated using CFX-Pre (Pre-Processor). The impeller with casing is schematically shown in Fig. 1. The impeller had an inner radius of 175 mm and outer radius of 250 mm. The blade inner and outer angles were 115^0 and 15^0 respectively. The chord length of the blade was 83 mm and there were 30 blades. The width of the impeller was 140 mm. The nomenclature used in the present study is shown in Fig. 1. The present problem has multiple frames of references, consisting of rotating domain (impeller) and the stationary domain (casing). Unstructured mesh was used for grid generation. Based on grid independency studies, the optimum mesh had 11,50,000 elements. Turbulence is modeled using RNG k-ε model. This model was chosen as it significantly improves the responsiveness to the effects of rapid strain and streamline curvature. As the flow in the CFF is unsteady, a time accurate turbulent flow simulations is sought for the prediction of flow characteristics. This unsteadiness is taken into account by sliding mesh interface, using transient rotor - stator change. CFF can be stated as a fan producing velocity pressure; therefore, casing is a must to effectively convert this velocity head to static head. Hence casing design plays an important role in the design of CFF. In the present investigations, inflow angle, out flow angle, casing outlet area are systematically varied and finally a better CFF configuration is arrived at. In the present study, the casing modifications are done based on flow vectors, distribution of total pressure and on efficiencies of impeller and casing. The detailed modifications along with casing parameters are listed in Table 1. Figure 2 shows the axially mass averaged total pressure coefficient plotted along the circumference for various casing modifications. Unlike axial or centrifugal machines, the total pressure is not uniform along the circumference of the fan. This is why CFF is called unconventional turbomachine. As no energy is added to the fluid, total pressure at the inlet of the impeller is atmospheric for most configurations ($\theta = 0^o$ to around $\theta = 120^o$, Fig.2). Total pressure drops in the region of eccentric vortex. There after the total pressure increases up to about $\theta = 270^o$. In this region, the flow follows the blade curvature with minimum incidence losses. Hence the energy transfer is maximum. From around $\theta = 270^o$, the total pressure gradually drops to zero values as it approaches the inlet. The width of the peak, i.e. number of blade passages contributing to the energy transfer to the flow is gradually increasing as the casing modifications are performed and finally configuration C7 has the largest width of the peak indicating that more number of blades is participating in transferring energy to the fluid. The impeller efficiency has increased with each casing modification where as casing efficiency reached its highest efficiency for casing C5 and there after the efficiency of the casing drops (see Table 1). But taking overall efficiency of the stage, casing C7 is a good compromise between impeller efficiency and casing efficiency.

W-2G-2. COMBINATION OF ADAPTIVE GRID-EMBEDDING AND EQUATION ADAPTATION METHODS FOR COMPRESSIBLE VISCOUS FLOWS

S. A. A. MIRJALILY, M. AMERI, *Department of Mechanical Engineering, University of Shahid Bahonar, Kerman, Iran,* Various adaptation techniques for the computation of viscous flows have been employed. This work describes combination of adaptive grid-embedding and equation adaptation methods on semi-structured grids for compressible viscous flows. In this combination, Navier-Stokes equations are solved within appreciably viscous macro-cells, whereas for the remaining macro-cells the equations are reduced to the Euler equations. Meanwhile, subdividing is used in macro-cells from the beginning of the adaptation procedure without any restriction. This combination is used to solve two internal and external flow model problems. The combination of adaptive grid-embedding and equation adaptation is complex, but it is shown in this paper that it is efficient and worth to be used in CPU time and memory space consuming problems.

W-2G-3. NUMERICAL SIMULATION OF EDDY STRUCTURES INDUCED WITHIN A WEDGE BY A HONING CIRCULAR ARC

LIN Changsheng, *Nanjing Institute of Technology* (*NJIT*), *Institute of Nonlinear Physics* (*INP*), *China*, WANG Dianshun, *Jilin Agriculture Engineering Polytechnic College*, *China,* LIU Yangzheng, *NJIT*, *INP*, *China*, Recently, Hills exploit the Moffatt eddy functions as a finite basis in a numerical collocation scheme to resolve Stokes flow within a wedge by a honing circular arc. In this paper, we present a BEM to investigate numerically same problem. The idea of the BEM is based on a fundamental solution of Stokes equation as Green's function and to convert Navier-Stokes equations to nonlinear boundary integral equations. For the low-Reynolds-number problems, the nonlinear boundary integral equations are reduced to solving linear boundary integral equations, with the boundary values, the velocity and pressure fields in the flow domain can be simulated numerically. For example, the method has been applied to calculate the problems. One of the main task is studying numerical simulation of velocity fields, because of central importance in the study was to examine the effectiveness of the method. On the other hand, simulating the velocity field of the flow can be compared with Hills's numerical results. By comparing, the numerical results are in good agreement with Hills's ones that shown the method is very effective and reliability in researching and solving Stokes flows within an arbitrary wedge by a honing circular arc. The question of especial importance is the method not only is more suitable for the arbitrary flow of a viscous fluid near a shape corner but also can suitable study the pressure and wall shear stress on the boundary and in the flow region as well.

W-2G-4. A NUMERICAL ANALYSIS OF FLOW AND HEAT TRANSFER IN THE DIMPLE PLATE HEAT EXCHANGER

Hyuk Jin AHN, *Sogang University, Korea*, Sang Hyuk LEE, *Sogang University, Korea*, Nahmkeon HUR ,*Sogang University, Korea,* In this study, the characteristics on the internal flow and heat transfer of the dimple plate heat exchanger were numerically investigated. For the numerical analysis, the conjugate heat transfer method between hot fluid - plate - cold fluid was used with appropriate boundary conditions. Based on these conditions, the characteristics on the flow and heat transfer of the heat exchanger were obtained. These numerical results were validated by the comparison with the experimental data. From these results, the correlations of the j-factor for the heat transfer and the f-factor for the flow friction were obtained for the present dimple plate heat exchanger. Furthermore, to enhance the performance of the heat exchanger, the effect of the ratio of the channel height to dimple diameter on the heat transfer and flow friction was studied.