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.