$Y=A*\sin(2*\pi*f*t)$, the first and the second element will flap passively. The velocity is set to zero initially and a non-slip boundary condition is applied on the four sides of the computational domain. As that indicated in the paper by Childress et al, the lift production from this 'hovering simulator' is associated with the unsteady aerodynamics and will be determined by the strength of vortices generated near the wing tip during flapping. For an oscillating rigid body motion (with infinite large stiffness and no flapping), the upward and the downward forces that the fluid acts on the flapper are equal. However, if the two elements can flap (with smaller spring stiffness of Ks=50), lift is produced when the body oscillates in the fluid. The lift force produced in one period is found to be 20 through time integration. We also study the effect of stiffness, amplitude and frequency on the lift production at a fixed Re number. A smaller stiffness results in large flapping angle and larger lift. Larger oscillating amplitude enhances lift production. Frequency can enhance the downward force and the upward force, but large frequency can not generate big lift remarkably.

10:40-12:00 (Room103)

Flow Stability (II)

Session Chair : Prof. C. F. Li, Jiangsu Univ/China

W-2C-1. STABILITY OF INVISCID INCOMPRESSIBLE PARALLEL CHIRAL FLUID FLOW IN THE PRESENCE OF TRANSVERSE MAGNETIC FIELD

K. P. SHIVAMURTHY, *Electrical and Electronics Engineering Department, Siddaganga Institute of Technology Tumkur*, *India,* G. K. SURESH, *Telecommunication Engineering Department, Siddaganga Institute of Technology, Tumkur*, *India,* N. RUDRAIAH, *UGC-Centre for Advanced studies in Fluid Mechanics, Department of Mathematics, Bangalore University, India,* In these days of globalization, modernization and liberalization, one has to become globally competitive. In this direction, material science plays significant role because of its potential applications in minimizing the weight and maximizing the performance. Therefore, one has to think of new materials like nano, smart, chiral materials and so on. Chiral material is defined as the one which cannot be brought into congruence with its mirror image by translation and rotation. It has the property of either left handedness or right handedness. Fluids like turpentine, sugar solutions, many organic compounds and body fluids possess chirality property. At present literature is available on the study of solid chiral materials but much attention has not been given to chiral fluids in spite of their immense use in biomedical engineering, chemical industry and so on. These applications require the effect of Lorentz force due to applied magnetic field and convective current, supplemented with the usual Maxwell's equations incorporating the chirality coefficient. To understand the physics of this steady chiral fluid flow and its use in the design of antenna in the presence of an external constraint of magnetic field, it is essential to study the stability of the flow to know under what condition this steady flow is stable, unstable or neutrally stable. This aspect has not been given any attention to our knowledge and hence it is investigated in this paper using the energy method supplimented with Galerkin technique. Considering charge stratification either increasing or decreasing, we have shown that in chiral fluid even in the absence of Rayleigh's inflexion point the chirality introduces instability depending on the value of the electromagnetic number. This criterion has been improved following the work of Fjortoft. A semicircle theorem is also proved to show that the frequency for unstable motion lies in that semicircle.

W-2C-2. ELECTRICALLY CONDUCTING FLOW PAST AN IMPERMEABLE SPHERE EMBEDDED IN A SPARSELY PACKED POROUS MEDIUM IN THE PRESENCE OF TRANSVERSE MAGNETIC FIELD

D. V. CHANDRSHEKHAR, *Department of Mathematics, Vivekananda Institute of Technology, Gudimavu, Kumbalagodu, India*, N. RUDRAIAH, *National Research Institute for Applied Mathematics (NRIAM), India and UGC-CAS in Fluid Mechanics, Department of Mathematics, Bangalore University, India,* The present day trend in Industrial and population growth have severely strained in providing uninterrupted power supply in the developing countries. The International atomic energy agency (IAEA) is proposing to overcome the shortage of power by using Inertial Fusion Energy (IFE), one of the unconventional and atmospheric friendly energies obtained by fusing the Deuterium – Tritium (DT), in IFE target which is of the form of sphere. Symmetry of the IFE target is one of the mechanisms to increase the efficiency of the IFE. The process of fusing DT to overcome an attractive force between them causes asymmetry of the target. To increase the efficiency of IFE it is essential to reduce this asymmetry of IFE target. At present, Researchers have proposed mechanisms like gradual variation of density instead of abrupt change, a foam layer at the ablative surface to reduce the asymmetry of the target. The objective of the present paper is to propose the use of retarding nature of Lorentz force generated by the motion of an electrically conducting DT fluid past an impermeable sphere embedded in a sparsely packed porous medium in the presence of a traverse magnetic field, using the Darcy – Brinkmann equation. A closed form exact solution is obtained for tangential and normal components of velocity and separation parameter. These are numerically computed for different values of c, the sum of porous parameter and Hartman number. We found that an increase in c is to decrease the thickness of the boundary layer which is favorable to reduce the asymmetry of the target and hence facilitates to increase the efficiency of IFE. The explanation of the velocity overshoot behavior is also given. The effect of shear stress and the separation parameter required to maintain the spherical symmetry are also discussed for different values of c.

W-2C-3. LASER DOPPLER ANEMOMETER (LDA) MEASUREMENT OF THE FLOW IN BETWEEN THE IMPELLER BLADES OF A CENTRIFUGAL BLOOD PUMP

L. P. CHUA, G. L. SONG and T. M. LIM, *School of Mechanical & Aerospace Engineering, Nanyang Technological University, Singapore,* During development of a rotary cardiac assist pump, sufficient understanding of the pump inner flow must be achieved so as to optimize the pump property in hemolysis reduction and thrombus prevention. In previous studies, a 5-time scaled up model of the Kyoto-NTN blood pump has been built according to flow similarity law and the flow in between the impeller blades has been measured using laser Doppler anemometer (LDA). In the present study, the flow between the impeller blades was measured directly on a 1:1 pump model using LDA in order to clarify the scaling effect on the flow. In the measurements, the impeller rotating speed was fixed at 2000 rpm and the flow rate was 5 l/min according to the pump working condition. A working fluid matching with the refractive index of the acrylic plastic (RI=1.49) was used to improve laser signal. Through the comparison between the present measurement result on the 1:1 pump model and those from the enlarged pump model, it can be found that there are remarkable differences between the flow patterns in the two pump models. The differences should be due to the incomplete similarity between the 5:1 pump model and the prototype. The present study indicates that the incomplete similarity of the enlarged pump model could produce remarkable different flow pattern. Therefore, it is emphasized that a complete flow similarity should be achieved when study of the blood pump is carried out on an enlarged pump model. Otherwise, considering the difficulty in achieving the complete similarity between the enlarged pump model and prototype, studies should be carried out using an exact size pump model as the prototype of the enlarged pump model.

W-2C-4. CHRARCTERISTICS OF VERTICALLY INJECTED BUOYANT JETS OF HIGHLY DILUTED PROPANE

K. W. CHUN, *Seoul National University, Korea,* J. KIM, *C.N.R.S. et Ecole Centrale Paris, France,* S. H. CHUNG, *Seoul National University, Korea,* In coflow jets with the nozzle diameter of O (1 cm) and the fuel jet velocity of O (10 cm/s), the buoyancy induced by the density difference between the fuel and air influences the jet structure appreciably. The present study investigated the behavior of such a buoyant jet numerically and experimentally, especially when the fuel stream had higher density than air. When the fuel jet was composed of propane highly diluted with nitrogen, the fuel jet was decelerated and formed a stagnation region. Consequently, the fuel was carried downstream by the coflow having a circular cone shape. When the fuel was moderately diluted or as the jet velocity increased, numerical results showed the Kelvin-Helmholtz type instability along the mixing layer of the jet. When the fuel jet velocity was relatively high, the stagnation height increased nonlinearly with fuel jet velocity having the power of approximately 1.62. In the relatively high Reynolds number regime of $Re > 80$, the stagnation height can be correlated to $Re^{0.62} Ri^{-0.5}$, indicating the combined effects of buoyancy and jet momentum. As the Reynolds number becomes small, the stagnation height was affected by the streamwise diffusion due to fuel concentration gradient and by the wake behavior near the nozzle tip. Accordingly, the stagnation heights approach to none-zero values, which were found to be relatively insensitive to fuel dilution.

10:40-12:00 (Room104)

Aerodynamics (IV)