Wednesday, August 20

09:00 ~ 10:20 (Room101)

Flows with Heat Transfer (II)

Session Chair : Prof. J. S. Park, Halla Univ/Korea

W-1A-1. HEAT TRANSFER ON TWO HEATED NEARBY CIRCULAR CYLINDERS

T. HAN, K.-S. YANG, D.-H. YOON, K. LEE, *Inha University, Korea,* Forced convective heat transfer around a group of cylinders immersed in crossflow is often found in many practical engineering applications. In designing such applications, it is essential to understand interference among adjacent wakes in heat-transfer characteristics as well as flow fields to achieve high efficiency in their performance. Heat transfer on two nearby circular cylinders of equal diameter immersed in the uniform crossflow at *Re*=120 and *Pr*=0.7 was numerically studied. We consider all possible arrangements of the two cylinders in terms of the distance between the two cylinders and the inclination angle with respect to the direction of the main flow. It turns out that significant changes in the characteristics of heat transfer are noticed depending on how they are positioned, resulting in quantitative changes of heat transfer coefficients on both cylinders. Collecting all the numerical results obtained, we propose a contour diagram for averaged Nusselt number for each of the two cylinders. The perfect geometrical symmetry implied in the flow configuration allows one to use those diagrams to estimate heat transfer rates on two identical circular cylinders arbitrarily positioned in physical space with respect to the main flow direction.

W-1A-2. EXPERIMENTAL AND NUMERICAL STUDY OF NATURAL CONVECTION FROM A FREE STANDING HEAT GENERATING ELEMENT IN A VERTICAL CHANNEL

M. R. RAJKUMAR, *Collegeof Engineering, Kerala, India*, S. ANILLAL, *Collegeof Engineering, Kerala, India,* This paper presents the experimental and numerical results of steady laminar convection of air between two vertical plates due to heating from a hot plate placed between them. The parallel plates as well as the central hot plate are polished to reduce heat transfer by radiation.. Experiments were conducted for aspect ratio 1.4 with heat source placed at the bottom. In the numerical investigation, laminar two dimensional steady state conditions are assumed, geometry is simulated, volumetric heat generation is given to simulate the convective Nusselt number.. An extended computational domain was adopted, which allows taking into account the diffusion by momentum and energy outside the channel. The Nusselt number obtained from experiment is compared with numerical value.

W-1A-3. EXERGETIC ANALYSIS OF VAPOUR ABSORPTION REFRIGERATION SYSTEM

Lijo VINCENT, H.D.KIM, *School of Mechanical Engineering, Andong National University, Andong, Korea,* RAJESH G., *College of Engineering, Thiruvananthapuram, Kerala, India,* John M. GEORGE, *T.K.M College of Engineering, Kollam, Kerala, India,* In this study, the first and the second law of thermodynamics are used to analyze the performance of a vapour absorption refrigeration system. A mathematical model based on the exergy method is introduced to evaluate the system performance and exergy loss of each component. Parameters connected with performance of the cycle– circulation ratio, coefficient of performance, and exergetic efficiency are calculated from the thermodynamic properties of the working fluids at various operating conditions. From the analysis performed, the suitability of five different fluid pairs are examined from the viewpoint of minimum total exergy loss. Using the developed model, the effect of main system temperatures on the performance parameters of the system and exergy loss of each component is analyzed in detail. It is found that with a decrease in the condenser temperatures, and an increase in the generator temperatures, the performance of the system increases. Compared to other components the generator heat transfer rate is the highest and the solution pump power is the lowest. Thus the effect of the pump on the total energy inputs is negligible. Moreover, the heat transfer rate of the refrigerant heat exchanger is lower than that of the solution heat exchanger due to lesser mass flow rate and temperature difference between the fluids. Exergy losses in the expansion valves, pump and heat exchangers are small compared to that in other components. The exergy destructions are significant in generator and absorber, which therefore make the generator and absorber the most important components of the cycle.

W-1A-4. THE CONSTANT TEMPERATURE ON BOUNDARIES FOR NATURAL CONVECTION IN A NARROW HORIZONTAL CYLINDRICAL ANNULUS

Kamyar MANSOUR, *Department Of Aerospace Engineering And New Technologies Research Center Amir Kabir University of TechnologyTehran, Iran,* We consider the two-dimensional problem of steady natural convection in a narrow Horizontal Cylindrical annulus filled with viscous fluid due to constant temperature on the outer and at the inner boundaries. The solution is expanded in powers of a single combined similarity parameter introduced by [1], which is the product of the Gap ratio to the power of four, and Grashof number $K = \varepsilon^4 G$ and the series extended by means of symbolic calculation up to 16 terms. Analysis of these expansions allows the exact computation for arbitrarily accuracy up to 50000 figures. Although the range of the radius of convergence for the similarity parameter K is almost zero but Pade approximation lead our results to be able to go as far as K=1500 and that is real success.

[1] Mansour, K. 1993 Using Stokes Expansion For Natural Convection inside a two-dimensional cavity. *Fluid Dynamics Research*, 1-33.

09:00-10:20 (Room102)

Biofluid Dynamics (I)

Session Chair : Prof. K. Nagayama, Kyushu Tech Univ/Japan

W-1B-1. PARTICLE SIMULATIONS OF THE DEFORMATION OF RED BLOOD CELLS IN A CAPILLARY VESSEL

K. NAGAYAMA and K. HONDA, *Kyushu Institute of Technology, Japan,* With the increase in arteriosclerosis, thrombosis, etc., in order to find out the cause, research of the flow characteristic of blood attracts attention. As for the analysis of the flow phenomenon of the RBC (Red Blood Cell or Erythrocyte), numerical analysis as well as experimental observation, is becoming a strong tool. Particle methods treat both solid and liquid as particles, and can be applied to complicated flow analysis. When applying a particle method to the flow analysis of RBC, RBC is divided into the elastic film and internal liquid. The RBC which is actually flowing in our body occupies 40-60% by volume ratio of blood (Ht: hematocrit). The objective of our research is clarifying the flow characteristic of the blood flow considering interactions RBCs and vein. In this paper, particle model is applied to the capillary straight tube flow. Deformations of RBCs in various ID (inner diameter) capillaries and hematocrit were studied. In case of ID=5.5 m Ht=0.31, RBC flows in lines contacting with the wall and deforms to consistently non-axisymmetric rocket shape. In case of ID=8.5 m Ht =0.2, RBCs flow at center of the vein, parachute type deformation was observed. In case of ID= 7.37 m Ht = 0.49, RBC interacts (multi-file flow) with each other and contact with the wall, forming zipper shape. In case of ID=8.7 m Ht =0.54, RBC interacts (multi-file flow) strongly with each other and contact with the wall, forming strong and complex deformation. Transition from single-file to multi-file flow as a function of hematocrit in capillaries of various diameters is studied. RBCs are single-file in narrow tube and at low hematocrit, while they are multi-file as the tube diameter increases or hematocrit increases.

W-1B-2. HYDRODYNAMIC PROPULSION OF A FLEXIBLE FOIL

Mysa Ravi CHAITHANYA, Kartik VENKATRAMAN, *Aerospace Engineering Department Indian Institute of Science, Bangalore, India,* A numerical analysis of the propulsive characteristics of a chord wise flexible foil in a potential flow is presented. The objective is to understand the effect of inertial and elastic effects on propulsive efficiency and thrust coefficient. The time dependent pressure loading, free stream velocity, and local displacement, velocity and acceleration of the hydrofoil, determine instantaneous response. That is to say, the dynamic coupling between the hydrofoil and surrounding fluid determines the lift and thrust force acting on it. The hydrofoil is prescribed sinusoidal heave motion. The literature on oscillating foil propulsion clearly points to the fact that flexibility of the fish body together with its inertia, and its effect on propulsion, has not been studied. The present work incorporates inertial loading due to the prescribed motion of the foil, and more importantly, includes flexible body dynamics in computing the unsteady fluid forces. A non-dimensional governing equation of the dynamics of fluid-structure interaction is derived. The two-dimensional potential flow dynamics is solved using a panel method. The flexible beam dynamics is numerically solved using a finite element method. These equations are numerically integrated in time simultaneously. The results are significant in that the mass ratio and characteristic natural frequency of the flexible foil seem to have a significant effect on the propulsive performance of the flexible foil. It is shown that thrust coefficient and propulsive efficiency is high for lighter and flexible foils.