flow characteristic in AAA is closely related with the rupture of aneurysm. The wall shear stress has been considered to influence the formation, growth, and rupture of AAA. On this account, it is very important to understand the flow features in the aneurysm. In this study, the velocity fields inside a typical AAA were measured using a transparent RP (rapid prototype) model under the pulsatile flow condition. Velocity fields were measured at different pulsatile phase angles using a PIV (particle image velocimetry) system. A large-scale vortex was formed inside the AAA. Vortices located near the wall of the AAA seem to increase the local pressure and wall shear stress. The AAA wall stresses are one of the most important governing factors contributing to the ruptured aneurysm.

W-3B-2. HEMODYNAMIC ANALYSIS OF PULSATILE BLOOD FLOW IN THE ARTERIAL BIFURCATION CASCADE OF A CHICKEN EMBRYO

J. Y. LEE, *POSTECH*, *Korea*, S. J. LEE, *POSTECH, Korea,* The arteries are very important in cardiovascular system and easily adapt to varying flow and pressure conditions by enlarging or shrinking to meet the given hemodynamic demands. The blood flow in arteries is dominated by unsteady flow phenomena due to heart beating. In certain circumstances, however, unusual hemodynamic conditions cause an abnormal biological response and often induce circulatory diseases such as atherosclerosis by inflammation. Therefore quantitative analysis of the unsteady pulsatile flow characteristics in the arterial blood vessels, especially arterial bifurcations plays important roles in diagnosing these circulatory diseases. In order to verify the hemodynamic characteristics, *in vivo* measurements of blood flow inside the extraembryonic arterial bifurcation cascade of a chicken embryo were carried out using a micro-PIV technique. To analyze the unsteady pulsatile flow temporally, the flow images of RBCs were obtained using a high-speed CMOS camera at 250 fps with a spatial resolution of 14.6 μm × 14.6 μm in the whole blood vessels. The variation of flow characteristics strongly depends on the vessel parameters. The mean velocity in the arterial blood vessel was decreased and pulsatility estimated by FFT analysis of velocity data extracted in front of the each bifurcation was also decreased as the bifurcation cascaded.

W-3B-3. EFFECT OF ANGLE ON HEMODYNAMICS OF PROXIMAL ANSTOMOSIS OF CORONARY ARTERY BYPASS GRAFTING

CHUA Leok Poh and Ji WENFA, *School of Mechanical & Aerospace Engineering, Nanyang Technological University, Singapore,* Bypass graft failure is a significant clinical problem and is frequently due to early postoperative graft thrombosis and eventual formation of intimal hyperplasia (IH). Hemodynamics is believed to play an important role in the onset and development of intimal hyperplasia. This study is designed to investigate the effect of anastomotic angle on the flow field of the proximal anastomoses, with emphasis on identifying site-specific hemodynamic features that could reasonably be expected to trigger the initiation and further development of IH. Five models including 30º, 45º, 60º, 75º and 90º models were investigated in the study. PIV measurement revealed that the flow field in the proximal anastomosis was strongly influenced by the anastomotic angle. Under pulsatile flow condition, large size of low recirculation flow was found along the graft inner wall just after the heel and decreased in size with decreasing of graft angle except the 30º model. Notable movement of the location of stagnation point at the graft outer wall was found at all models except the 90o model. Hemodynamic parameters including wall shear stress (WSS), spatial wall shear stress gradient (WSSG), time-averaged WSS (TAWSS), time-averaged WSSG (TASWSSG) and oscillating shear index (OSI) were derived. Regions of low-WSS-high-OSI and high-WSS-low-OSI were found around the anastomotic joints. The 45º model has the smallest size of such region whereas the 90º model has the largest one. To conclude, the 45º anastomosis model would provide the best graft patency rates among the five models investigated.

W-3B-4. BLOOD FLOW CHARACTERISTICS AND RBCS' MOVEMENT IN A MICRO-STENOSIS

H. S. JI, *POSTECH, Korea*, M. J. KANG, *Seoul Central Technology Appraisal Institute, KIBO Technology Fund, Korea,* S. J. LEE, *POSTECH, Korea*, The blood flow characteristics and movement of RBCs passing through a microstenosis have been considered to be closely related with circulatory disorders, one of the major causes of death in modern society. In this sense, the flow characteristics, especially the wall shear stress in the stenotic region have received large attention in recent decades. The hemo-rheological parameters, such as viscosity, hematocrit, deformation, shear rate and aggregation of RBCs, influence on the blood flow in a microvascular network. Microcirculation is very important for metabolism for a mammal body. However, most previous studies on the hemorheological characteristics of blood samples in a microstensis focus on the clinical point of view. Therefore, the flow characteristics of blood flow and motion of RBCs in the micro-stenosis were experimentally investigated using a micro-PIV technique. To simulate a blood flow related with arteriosclerosis, in vitro experiments were carried out using a microchannel with a micro-stenosis. The micro-PIV system consists of an inverted microscope, a double-pulsed Nd:YAG laser, a 12 bit cooled CCD camera, a delay generator, and a personal computer for control and data processing. The backlight method was employed to improve the image quality by reducing non-uniform illumination. A PDMS microchannel having a micro-stenosis with a severity of 80% was used as the experimental model of stenotic blood vessel. The width of straight channel and stenotic throat are 100 and 20, respectively. The depth of the microchannel is 50. Human blood donated form a healthy male donor was first heparinized to prevent coagulation and the blood samples were pre-treated to prevent biochemical interaction with fluorescent particles and blood samples. The fluorescent particles of 1.0 in a mean diameter were used for in-vitro micro-PIV experiments. Human blood with a 5% hematocrit was supplied into the micro-stenosis channel using a syringe pump. The flow characteristics and movements of RBCs through the micro-stenosis were investigated with varying flow rate. The same experiments were repeated in a straight microchannel under the same flow conditions to compare the flow characteristics in the micro-stenosis.

13:20 ~14:40 (Room103)

Turbulence Modeling

Session Chair : Prof. C. X. Xu, Tsinghua Univ/China

W-3C-1. A STUDY ON TVC USING TWO EQUATION TURBULENCE MODELS

V. NANDAKUMAR, P. SELVAGANESH and S. VENGADESAN, *Department of Applied Mechanics, Indian Institute of Technology Madras, Chennai, India,* Flow stabilization in combustors can be achieved by a novel method which employs a vortex that is trapped inside a cavity referred as Trapped Vortex Combustor (TVC). The cavity is formed between a forebody and an afterbody mounted in tandem as shown in Fig. 1. The combustor configuration chosen is the one that was used earlier for numerical investigations. Numerical investigation of flow fields for both non-reacting (cold flow) and reacting flow is performed. This involves (i) passive flow through TVC to obtain an optimum cavity size to trap stable vortices inside the cavity, (ii) effect of injection of fuel and air directly into the cavity, (iii) fuel/air mixing properties inside the cavity and (iv) effect of annular flow on reaction characteristics. Commercial CFD software FLUENT is used for this study. The main objective is to use two equation turbulence models (k-ε and k-ω models) for numerical investigation of TVC. Modified k-ω[1] and Non-linear k-ω[6] turbulence models are incorporated through User Defined Functions(UDF). For the reaction flow analysis a single step global chemical mechanism for methane-air combustion is employed. Combustion chemistry is handled by Eddy Dissipation model where reaction rates are assumed to be controlled by the turbulence and hence Arrhenius chemical kinetic calculations are avoided.

W-3C-2. A NOVEL MODEL BASED ON TURBULENT FLAME MODEL FOR SIMULATION OF TURBULENT INTERFACIAL FLOWS

E. SHIRANI, *IUT, Iran*, F. GHADIRI, *IUT*, *Iran,* In this work we have used Reynolds averaged 2D Navier-Stocks along with averaged volume of fluid advective equations based on volume of fluid to simulate turbulent interfacial flows. We have introduced a novel model for mean fluctuation of the volume of fluid-velocity correlation term based on the idea used for modeling turbulent flame front tracking model. In that model, the flame front-velocity correlation term was modeled and the normal gradient of the flame front was neglected. Here we show that for turbulent interfacial flows between two immiscible flows, this term play crucial role and have to be included in the model. To show the accuracy and capability of the model, the 2D K-H instability of high Reynolds number, as well as turbulent plane jet of water in still air was simulated and compared with experimental results. The model constant is σ_f and its order is examined for both of the simulated conditions. It was shown that the model simulate the flow with good degree of accuracy.

W-3C-3. PERFORMANCE ANALYSIS OF EDDY-VISCOSITY

BASED TURBULENCE MODELS FOR ATTACHED AND SEPARATED FLOWS

D. S. KULKARNI, B. N. RAJANI and S. MAJUMDAR, *CTFD Division, NAL(CSIR), Bangalore, India*, B. RAGHAVENDRA, *Department of Mechanical Engineering, Sir MVIT, Bangalore, India,* RANS (Reynolds Averaged Navier-Stokes) solvers coupled to appropriate eddy viscosity based turbulence models have emerged as the most cost-effective approach today for CFD (Computational Fluid Dynamics) analysis of industrial turbulent flow problems. Unfortunately there is no single reliable turbulence model which is universally applicable for any flow whatsoever. A common disadvantage of most of the widely used two-equation turbulence models is the inaccurate prediction of the near wall flow behaviour where the normal stress anisotropy arising out of the physical blocking of the wall-normal fluctuation cannot be captured by the simplified scalar eddy viscosity approach, even using the exponential damping functions in the near wall zones. The other important mean flow condition for which the popular two-equation turbulence models are often observed to be inadequate and inaccurate is the adverse pressure gradient and eventually flow separation. While considering the performance of a turbulence model, the main challenge is how accurately the model can capture flow separation in wall bounded turbulent flows– the inception of separation, reattachment and post separation recovery of the flow field. The present study is aimed at calculation of three different wall bounded two-dimensional turbulent flow situations, viz., fully developed flow through a plane channel, flow through a plane asymmetric diffuser and flow past an airfoil by solving Reynolds Averaged Navier Stokes (RANS) equations coupled to different eddy-viscosity based turbulence models. The turbulence models tested in the present study are low Reynolds number version of the k-ε model of Chien , k-ω model of Wilcox, Shear Stress Transport model of Menter as a weighted combination of k-ε and k-ω model and the simplified second moment closure based $k-\varepsilon-\overline{v^2}-f$ (so-called V2F model) model of Durbin. The relative performance of these different turbulence models is assessed through comparison of the computation results to corresponding DNS and/or measurement data, available in unclassified literature.

W-3C-4. NUMERICAL SIMULATION OF IONIZED PARTICLE MOTION IN FLUID FLOW

Y. KOJIMA, Y. ENDO, K. SAKABE, H. ISHIKAWA, *Tokyo University of Science*, *Japan*, T. SETO, *Kanazawa University*, *Japan,* For the purpose of contribution to the reduction technology of nanometer-scale chemical pollutants, DIMA was developed to measure and classify such nanometer-scale particles. DIMA has measuring principles based on differential mobility analyzer (DMA), with various improvements given to measure the nanometer-scale particles and ion clusters with high resolution and high sensitivity. Especially, DIMA made possible to measure the ion cluster. The DIMA analyzing region has a coaxial cylinder structure composed of an outer housing and an inner center rod. Nanometer-scale particles are ionized by electric charge, introducing into the analyzing region in inside of the DIMA with a carrier gas. In the analyzing region, ionized particles are forced to flow downward with clean sheath gas. The inner center rod is charged with high voltage and the outer housing is grounded, creating an electric field to attract ionized particles to drift toward the center rod. The particle path in the analyzing region depends on ion-mobility, and by giving a certain voltage of the center rod, it allows specific particle to correct from a sampling slit on the surface of the center rod. However, measurement error sometimes occurs by the disorder element which inheres in the internal flow of fluid drag, turbulence and the influence of Brownian motion. Therefore, it is important to explain ionized particle motion in flow field by numerical simulation. This paper proposes a three-dimensional numerical model to explain such motion inside the DIMA, with electric field, fluid drag, turbulence, Brownian motion, and repulsion taken into consideration. Three dimensional computational domain where is a portion of one fifth of DIMA coaxial analyzing region for the circumferential direction was used in this study. Ionized particle are introduced from an inlet slit which is located at the upstream outer housing wall. The width of inlet slit is denoted by "W". The bottom wall represents the surface of inner center rod which is charged with high voltage. An outlet slit, at which is located downstream of 100 times the width of the inlet slit, is used to capture ionized particle. In this study, ionized particle motion was calculated by overlapping the effect of flow field and the electric field. First, flow field was determined by solving the governing equations of the Navier-Stokes and the continuity equation under the assumption that the internal flow of the DIMA to be incompressible and laminar. The one-way coupling method was applied to disregard the effect of ionized particle motion on flow. The Reynolds number based on the flow velocity and inlet width of the sheath gas and the kinematic viscosity is in the range of 3,000 to 18,000. The ion cluster model was not taken in consideration in this study. From the result of the comparison of repulsive force with electric field, it is found that the repulsive force enhanced the dispersion of ionized particle motion. As for the dispersion of captured ionized particles, the effect on differences of ionized particles motion becomes dominant in order of repulsion, Brownian motion, turbulence and electric field.

13:20 ~14:40 (Room104)

Aerodynamics (V)

Session Chair : Prof. M. Takao, Matsue National College of Tech/Japan

W-3D-1. COMPUTATION OF FLOW FIELD AROUND RE-ENTRY CAPSULE AT SUPERSONIC SPEEDS

S. DAS, *Department of Space Engineering & Rocketry, B.I.T Mesra - Ranchi, India,* J. K. PRASAD, *Department of Space Engineering & Rocketry, B.I.T Mesra - Ranchi, India,* Re-entry missions are being thought upon as alternative to conventional launch vehicles and winged body due to various advantages. A reentry capsule will have the capability of vertical landing with advantages of more space for experimental payloads / instruments, brought back to earth, after completing the space missions. The blunt configurations are adopted to decelerate the vehicles for safer missions of return to earth. Reentry capsules generally exhibit aerodynamic instabilities due to formation of wake and possible flow separation at low altitudes, which may be detrimental for the mission. In general, flow field around a typical blunt capsule at supersonic speeds are characterised by presence of a strong detached bow shock, expansion at the shoulder, and a separated boundary layer at the rear surface. The separation depends on freestream Mach number, angle of incidence, Reynolds number etc. This shows the presence of complex flow structure around capsule with different flight conditions. The characterisation of flow field in the wake of the capsule is also important with a view of aerodynamic stability of the capsule. Numerical simulations have been made to obtain the flow field over a blunt body reentry capsule at supersonic speeds. Experiments involving schlieren and measurement of static pressures were made at a Mach number of 2. Comparison indicates good agreement between experiments and the computation. The flow around the reentry capsule has been computed at Mach numbers of 2, 3 and 5. Existence of a complex flow in the wake is observed which is characterized by re-circulating flow, separation zone, re-compression, vortex core and stagnation points. The change in wake flow field at lower Mach number is more predominant in comparison to higher Mach numbers which might be responsible to affect the overall aerodynamic parameter of the capsule.

W-3D-2. THE INFLUENCE OF REYNOLDS NUMBER ON THE AERODYNAMIC CHARACTERISTICS OF AIRFOILS WITH TRIANGULAR SHAPED THROUGH DAMAGE

Bahareh YAHYAVI, *K.N.Toosi University of Technology, Iran*, Masoud MIRZAEI, *K.N.Toosi University of Technology, Iran*, Mahmoud MANI, *Board Member, Center of Excellence in Computational Aerospace Engineering, Amirkabir University of Technology, Iran,* In this paper flow around a full span NACA 64_1-412 airfoil with two different orientation of triangle shaped damage was experimentally investigated using balance measurements and flow visualization. To assess the influence of Reynolds number on results, models of airfoil with 150mm and 100mm chord were used. Our studies were carried out in a low turbulence, closed circuit, wind tunnel at Aerospace engineering department of AmirKabir University of technology, with test section of 0.45×0.45 m. The experiments were done at air speed of 35 m/s. Each model was tested in both undamaged and damaged states. The standard tuft technique was used for flow visualizations. At any incidence, the increments are defined as: $dC_l = (C_l)_{damaged} - (C_l)_{undamaged}$, $dC_d = (C_d)_{damaged} - (C_d)_{undamaged}$, $dC_m = (C_m)_{damaged} - (C_m)_{undamaged}$ For damaged model, increasing incidence generally resulted in greater lift loss, higher drag and more negative pitching moment. Inverse triangle, in comparison with the right triangle case, resulted in the smaller drag increments and reduced lift increments. For the pitching moment increments the inverse triangle results are more negative than for the right triangle. Increasing incidence resulted in a stronger jet, with more significant effects on the flow over the airfoil. Changing the orientation of the damage to produce the inverse triangle resulted in significant changes in the flow characteristics. The main feature of these flows is identical to those previously described by Irwin for circular damage and also Mani & Render for triangular damage [3]. Our results indicated that the coefficient changes were dependent to the Reynolds number over the range tested, and by varying Reynolds number, the trends were largely changed. At lower Reynolds number, the rate of d[C_l] change increased with incidence dramatically, and the rate of d[C_d] change decreased over the full incidence