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