13:20 ~ 14:40 (Room101)

Supersonic and Hypersonic Flows (II)

Session Chair : Prof. H. Katanoda, Kagoshima Univ/Japan

W-3A-1. NUMERICAL AND THEORETICAL ANALYSIS ON DESTABILIZING MOTION OF SUPERSONIC CONFIGURATIONS

Y. J. YANG, *China Academy of Aerospace Aerodynamics, China*, E. J. CUI, *China Academy of Aerospace Aerodynamics, China,* W. J. ZHOU, *China Academy of Aerospace Aerodynamics, China,* The unsteady pitching motion of the supersonic axial-symmetric figure is simulated by the global sub-iteration in the fluid dynamic equation and the rigid dynamic equation. The numerical results show that the unstable free pitching of a flared axial-symmetry figure develops the limit-cycle motion in supersonic flow which is accompanied with unsteady structure. The flow physics of the self-oscillation includes the restoring mechanism of the static-stabile figure and the damping mechanism resulting from the shock wave flow hysteresis at the skirt section. Furthermore, the nonlinear dynamic equation in parameterizations is deduced from the second Lagrange equation and principle of virtual work, which can characterize the hysteresis. And the parameterized motion is approximately analyzed utilizing MTS (Multiple Time Scales) method. The self-oscillation is the quasi SHM (simple harmonic motion) and the static stability of the aircraft is necessary for such periodical motions. The damping at the equilibrium is the bifurcation parameter determining the dynamic stability. The amplitude is correlated with the nonlinear damping and the frequency is correlated with the nonlinear rigidity. The theoretical analysis and the numerical reconstruction, in regard to the nonlinear model embodied by parameter identification, agree well with the CFD results, which prove the modeling research valid.

W-3A-2. INTERACTION OF FORWARD AND AFT PITCHED PLATES ON BLUNTED CONE IN SUPERSONIC - HYPERSONIC FLOWS

Salimuddin ZAHIR, *PhD Candidate Aerospace Engineering, Northwestern Polytechnical University*, *Xi'an, PRC Member AIAA, China,* Zhengyin YE, *School of Aeronautics, Northwestern Polytechnical University, Xi'an, PRC, China,* Supersonic-Hypersonic flow interactions for short protuberances installed on a standard blunt cone configuration were studied, aerodynamic effects were found analogous to lateral jet-interactions for Mach 3.5 and 5.0 on a conic geometry at incidence. Static aerodynamic coefficients, axial and lateral pressure distributions were determined using CFD tools for flow interaction effects of pitched short protuberance geometries of cylindrical cross-section. It has been concluded that pitched short protuberance installed on a blunted cone causes an increase in net force through altering pressure distribution, with consequent development of aerodynamic pitching moment, forward pitching of protuberance was found to be more effective in comparison with an aft inclination, while similarity in predicted pressure distribution using CFD analysis with an overall prediction accuracy of ± 8% was found with the experimental results in the hypersonic range. Side thruster is a highly responsive means for attitude control with this background computational aerodynamic study using CFD analysis was conducted earlier; current work presented is with pitching of short protuberance forward and aft, to get a more realistic fluid flow situation, analogous to inclined jets, inferences presented here are for Mach 5 and 3.5 flows for study of static aerodynamic coefficients and axial pressure distributions for a blunted cone geometry for a fixed H/D tilted forward and rearward from its mean position. Single protuberance height equal to cylinder diameter was used and aerodynamic flow field behaviour for hypersonic free stream interaction with lateral short protuberance in pitched forward and aft positions were analyzed by calculating static coefficients, axial and lateral pressure distributions and flow visualization using pressure and velocity contours.

W-3A-3. COMPARATIVE STUDY OF STREAMWISE AND SPANWISE VORTICAL DISTURBANCES IMPOSED ON A HIGH SUPERSONIC FLOW OVER A CONE CYLINDER

Naresh KUMAR, *National Aerospace Laboratories, Bangalore, India*, T. K. SENGUPTA, *IIT Kanpur, Kanpur, India,* Receptivity represents first stage in the laminar to turbulent transition process. It is the mechanism by which the flow responds to the imposed disturbances. For high speed flows this process is complicated by the interaction these imposed disturbances with the shock waves present. The objective of the present work is to perform receptivity analysis of the flow over a cone-cylinder configuration at M = 4 for various freestream vortical disturbances. The interaction of these freestream disturbances with the oblique shock wave present and effect of this interaction in the region downstream is also studied. Computational Fluid Dynamics (CFD) based on DNS approach enables us to give some insight into transition process. The full three-dimensional unsteady Navier-Stokes (NS) equations are solved at a high Reynolds number using the high accuracy compact schemes (S-OUCS4 and OUCS4) (Sengupta, 2006) to resolve all the spatial and temporal scales. Since the computational size of such real problems to be solved with DNS is also very large, it is mandatory to have a DNS code that is highly efficient in terms of parallel computation. Parallelization is done using techniques given in (Sengupta, 2007). Validation of the present code is done against the experimental data (Stalling, 1980). Equilibrium flow is first computed using the NS equations and then the flow is perturbed by time periodic sources of vortical pulses placed at the free-stream, and same NS equations are used to study receptivity and further growth of disturbance. Essential idea is to bring out the difference between spanwise and streamwise type of vortical disturbances imposed over a hypersonic flow.

W-3A-4. NUMERICAL PREDICTION OF THE HEAT TRANSFER IN HYPERSONIC FLOW USING AUSM SCHEME

S. P. NAGDEWE, H. D. KIM, *Andong National University, Korea*, T. SETOGUCHI, *Saga University, Japan,* In recent years, scientific community has found renewed interest in hypersonic flight research. These hypersonic vehicles undergo severe aero-thermal environment during their flight regime. Simulation of hypersonic flow problems encounters all the complications. It has severe viscous dissipation in a boundary layer, strong shock waves and expansions, embedded subsonic regions, shock boundary layer interaction, and many more. In addition to this, there is the region of high temperature, real gas effects. Moreover, hypersonic flows have different characteristic scales and hence finer meshes, which require more computational time for simulation. Thus, there is a need to choose a numerical scheme, from amongst the available schemes, which is inherently robust at the same time less dissipative, accurate but efficient and lastly requiring less storage and less expensive. In present work, AUSM scheme has been selected for the simulation of hypersonic flows. One of the most important topics of research in hypersonic aerodynamics is to find reasonable way of calculating either the surface temperature or the heat flux to surface when its temperature is held fixed. This requires modeling of physical and chemical processes. Hyperbolic system of equations with stiff relaxation method are being identified in recent literature as a novel method of predicting long time behavior of systems such as gas at high temperatures. In present work, Energy Relaxation Method (ERM) has been considered to simulate the real gas flow over a 2-D cylinder. Computations have been carried out by using the finite volume based density solver. Present heat flux results over the cylinder compared well with the experiment. Thus, real gas effects in hypersonic flows can be modeled through energy relaxation method.

W-3A-5. SUPERSONIC COMBUSTION OF HYDROGEN INJECTED UPSTREAM OF A STEP-A NUMERICAL STUDY

M. DEEPU, *Dept. of Aerospace Engg; Indian Institute of Space Science &Technology Trivandrum, India*, S. JAYARAJ, *Department of Mechanical Engineering, NIT Calicut, India*, H. D. KIM, *Andong National University, Korea,* G. RAJESH, *Department of Mechanical Engineering, College of Engineering, Trivandrum, India,* Numerical simulation of supersonic combustion of hydrogen in air has been done using point implicit Finite Volume Method. This method treats all chemical species terms implicitly and all other terms explicitly. The developed solver is based on the solution of unsteady, compressible, turbulent Navier-Stokes equations, using Unstructured Finite Volume Method (UFVM) incorporating RNG based κ-ε two equation model and time integration using three stage Runge-Kutta method. An eight-step hydrogen-air finite rate chemistry model was used to model the reacting flow field. The preconditioning of chemical source terms is found to be effective in overcoming the stiffness. This code has been used for simulating the flow field resulting due to the interaction of sonic hydrogen jet injected upstream of step into supersonic cross flow. The effect of expansion wave resulting due to step in mixing and combustion is studied. The flame holding capability of step is also established.

13:20 ~14:40 (Room102)

Biofluid Dynamics (III)

Session Chair : Prof. J. H. Shin, KAIST/Korea

W-3B-1. IN VITRO ANALYSIS OF BLOOD FLOW IN AN ABDOMINAL AORTA ANEURYSM

J. P. LEE, *POSTECH*, *Korea*, D. S. KIM, Seoul Veterans Hospital, Korea, S. J. LEE, *POSTECH, Korea,* The complicated features of blood flow in the abdominal aorta aneurysm (AAA) are receiving large attention, because the 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