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