e-dimensional phenomenon which requires finite time. However, just it is shown qualitatively concerning effect of diaphragm rupture time. A s far as we know, there are no researches for quantitative and precise investigation of diaphragm rupture process. In the present research, t he numerical study was carried out in order to investigate the effect of the diaphragm rupture of the shock tube on the characteristics of expansion and shock waves generated near the diaphragm. Furthermore, the time-dep endent behavior of non-equilibrium condensation of moist air through the shock tube was investigated numerically. For simulations, the diap hragm of the shock tube was assumed to open in finite time and the opening rate of the diaphragm rupture was changed with time. Experi ments were also carried out in order to investigate the effect of the d iaphragm rupture process on the flow characteristics of expansion and shock waves. As a result, it was found that the simulated values agreed well with experimental results using a suitable equation showing the di aphragm rupture process, and the opening of the diaphragm in finite ti me has effects on the relationship between pressure and the position t hat condensate mass fraction begins to increase, and shock Mach num ber.

09:00-10:20 (Room104)

Aerodynamics (III)

Session Chair : Prof. M. H. Sohn, KAFA/Korea

W-1D-1. UNSTEADY RANS SIMULATION OF INCOMPRESSIBLE FLOW PAST A SYMMETRIC AEROFOIL AT HIGH ANGLES OF ATTACK

Sekhar MAJUMDAR, B. N. RAJANI, D. S. KULKARNI, M. B. SUBRAHMANYA, *Computational & Theoretical Fluid Dynamics Division National Aerospace Laboratories (CSIR), Bangalore, India,* Flow past two-dimensional aerofoils at high angles of attack beyond stall at moderately high Reynolds number, becomes essentially three dimensional and has strong non-linearities due to unsteadiness and flow separation. In the Unsteady Reynolds Averaged Navier Stokes (URANS) methodology, the governing equations for mean flow, coupled to an appropriate statistical turbulence model, are framed on the basis of phase averaging. The present work aims at prediction of unsteady flow past a stationary NACA 0012 aerofoil at a chord based Reynolds number of 1 million, for a wide range of angles of attack varying from 0^0 to 90^0. The computation uses an in-house-developed time-accurate RANS code, based on a pressure-velocity solution strategy, coupled to second order accurate numerical schemes for spatial and temporal discretisation of the convective fluxes and variety of eddy viscosity based turbulence models. A 2-block O-grid consisting of 320 × 100 control volumes has been employed with the far field placed at a radius of 30C and the minimum near wall distance of the first grid node is so chosen that the corresponding y^+ is less than unity. The computation shows the flow to be steady below the stall situation, whereas for high angles of attack, the mean flow attains a kind of periodic state with a dominant frequency. Typical instantaneous particle traces and vorticity contours at high angles of attack show the vortex street formation in the wake region. Reasonable agreement is obtained between the present prediction and the corresponding measurement data for the mean aerodynamic coefficients up to an angle of attack as large as 90^0. The sensitivity of the computation results on the turbulence models and the discrepancies with the measurement data may be attributed to the inherent inadequacy of any eddy viscosity based turbulence model in prediction of separated and transitional flows.

W-1D-2. WIND TUNNEL TEST FOR WIND TURBINE AIRFOIL

H. K. SHIN, *Korea Institute of Energy Research*, *Korea,* S. W. KIM, *Korea Institute of Energy Research, Korea,* In wind turbine blades, airfoils are required to have different characteristics as compared with airplane airfoil. Airfoils for wind turbine blade must have a high lift-to-drag ratio, moderate to stall behavior and especially, low roughness sensitivity. Also an operating Re. No.s are lower than conventional airplane airfoils. At mid-span and inboard region, structural problems have to be considered. Especially, for stall regulated type, moderate stall behavior is essential requirement. For these reasons, airfoil design for HAWT blade is essential part of blade design. Korea Institute of Energy Research (KIER) has designed airfoils set for 100kW stall regulated wind turbine. Then wind tunnel test was implemented to validate designed airfoils and analyze airfoil characteristics.

W-1D-3. A NUMERICAL INVESTIGATION ON DIFFERENT REGIMES OF DYNAMIC STALL AND MACH NUMBER EFFECTS ON DEEP STALL REGIME

S. FOTOVATI, H. EMDAD & R. KAMALI, *Department of Mechanical Engineering, School of Engineering, Shiraz University, Iran,* Dynamic stall is a phenomenon caused by vortex shedding on the upper surface of oscillating airfoils at high angle of attack. This causes decrease in lift and massive increase in drag force. Dynamic stall and unsteady boundary-layer separation have been studied in compressible turbulent flow. By varying the frequency and angle of attack of NACA 0012 airfoil, different types of stall were observed, and the vortex-shedding phenomenon was found to be the predominant feature of each. In all cases, the deeper dynamic stall was caused by increasing values of maximum incidence, existence of separated boundary layer on mean angles of attack and increase in reduced frequency. In addition, the effects of moderate subsonic Mach number near to transonic range were applied and the aerodynamic characteristics caused by this situation came into analysis. In this research, fully turbulent flow with free stream Reynolds No. of 2500000 considered. For analyzing different regimes of dynamic stall, free stream Mach number set equal to 0.3, but for observing the effects of changing in Mach number, different free stream Mach numbers, 0.09, 0.3, 0.55, 0.75 applied and the aerodynamic characteristics considered and came into discussion. It is expected by the Prandl-Gilauert Theorem that, lift coefficient drops while Mach number is increased.

W-1D-4. PARAMETRIC STUDIES ON A REAR FLOW SPOILER FOR ENHANCED STABILITY OF A SPORTS CAR

G. K. CHAITANYA, D. RAKESH, Q. H. NAGPURWALA and S. R. SHANKAPAL, *Department of Mechanical and Automotive Engineering, MSRSAS, Bangalore, India,* Numerical flow simulations have been performed on a typical sports car (SUV) to understand the inner working of the rear spoiler on the aerodynamic performance of the vehicle. Attention is paid only to two parameters, viz. spoiler orientation and height. The aim was to arrive at their possible optimum values for producing the desired negative lift with low drag. The geometric model (1:1) of the selected car was created using CATIA V5R15 software and the solver used was FLUENT with pressure based, segregated, implicit scheme and *k-epsilon* turbulence model. The CFD analysis was carried out for three vehicle speeds (inlet air velocities) of 80, 120 and 160 kmph, first on the baseline car configuration (without rear spoiler) and then on the same car fitted with the rear spoiler of aerofoil shape. The parametric studies were performed by changing the spoiler orientation from 0° to 20° in steps of 5°, and by increasing its vertical height by 25% and 50% of the mean vertical baseline height. From the baseline studies, it was found that the lift coefficient decreased by 28%, when a rear spoiler was installed at 0° orientation (chord horizontal). The lift coefficient further decreased with an increase in the orientation of the rear spoiler up to 10° and then it increased again. The best performance of the spoiler was obtained at 10° angular orientation and at an increased vertical height equal to 125% of the baseline height. These values were found to differ considerably from the default values for the existing spoiler in the car. It is shown that the presence of the spoiler brings about substantial changes in the flow structure around the spoiler and also behind the car, which are responsible for reduction in the net lift with slight increase in the vehicle drag.

09:00 ~ 10:20 (Room 105)

Compressible Flows (III)

Session Chair : Prof. M. Yaga, Ryukyu Univ/Japan

W-1E-1. AN EXPERIMENTAL STUDY ON TRANSONIC RESONANCE IN TWO DIMENSIONAL SUPERSONIC NOZZLE

S. J. JUNG, *Kyushul University, Japan*, M. YONAMINE, *Kyushul University, Japan*, T. AOKI, *Kyushul University, Japan*, H. D. KIM, *Andong National University, Korea,* The present paper describes an experimental work to investigate characteristics and generation mechanism of a transonic resonance in jet flow that is discharged from convergent-divergent nozzle. When the nozzle runs at a pressure ratio much lower than the design condition, the transonic resonance and tone are produced by the unsteady shock and acoustic resonance within the divergent section of the nozzle. Unfortunately, the exact generation mechanism is not completely understood yet. It has been known that the characteristics and origin of the transonic resonance are different from screech tone. For instant, the frequency of the transonic resonance due to the shock within the nozzle somewhat increases with an increase in the nozzle pressure ratio, and the staging behavior of the transonic tone is occurred by the odd-harmonic stages. The present work is accomplished in an anechoic test room. Acoustic tests show that the test room is anechoic for frequency components above approximately 120Hz and a background noise is about

10dB. Compressed dry air is stored in a high pressure tank that has a capacity of 5m^3, and is supplied to the plenum chamber, in which a honeycomb system reduces flow turbulence. A convergent-divergent nozzle with a design Mach number of 2.0 is installed in the end wall of plenum chamber. The nozzle has a throat height of 9.6mm, an exit height (*D*) 16.2mm, and a width 30mm. A Schlieren optical system is employed to visualize the qualitative structures of supersonic jet. Acoustic measurements are made using a condenser microphone that has a diameter of 6mm. The microphone is located at a 96° from the jet direction, and the radial distance of 34*D* (550mm) from the exit of the nozzle. The acoustic signals are analyzed by using a FFT analyzer. A FFT analysis provides the noise spectra, and providing the spectral data in the range from 0 to 40 kHz, with a frequency band width of 25 Hz. The temperature in the plenum chamber is measured by using a thermocouple (approximately 288K), and it maintains constant at room temperature during test. The pressure is measured by a pressure transducer flush mounted on the top wall of the plenum chamber and nozzle wall. The results obtained show that generation of transonic resonance is closely related to the pressure oscillations in the nozzle, and the large area of Mach stem increases the amplitude of transonic tone.

W-1E-2. COMPUTATIONAL ANALYSIS OF JET IN SUPERSONIC REGIME WITH VARIABLE AMBIENT CONDITIONS

Ajmal BAIG, S. BILAL and S. ZAHIR, *National Engineering & Scientific Commission, Islamabad. Pakistan,* Computational study of the behaviour of the supersonic jet in supersonic regime has been done for fully-expanded and under-expanded jets and compared with the still ambience. The supersonic jet is produced with a convergent-divergent nozzle. Initially the centre-line pressure distribution with an ambience of still air is calculated for fully and under-expanded jets and then the pressure distributions for fully and under-expanded jets are calculated when ambient flow is supersonic. The centerline pressure distribution in still air shows that the oscillations in fully expanded jet are very fast as compared to the under-expanded jet. These pressure oscillations are reduced in the supersonic regimes as a result of supersonic pressure relief effect. The length of shock cells for underexpanded jet is greater than the full-expanded jet both in still air and supersonic freestream. The computational analysis showed that the jet pressure adjust to the ambient pressure very quickly in the supersonic regime as compared to the still ambience. The problem is solved using CFD software based on the finite control volume technique. Air is taken as working fluid and solved as ideal gas. Viscosity is calculated using Sutherland law. For 2D axisymmetric geometry, the continuity and momentum conservation equations in axial and radial direction are solved. The energy equation is also solved. Density based solver with implicit formulation and 2nd order upwind flow discritization is used. Two equations k epsilon turbulence model with standard wall functions is used. Symmetry boundary condition is applied along the axis of symmetry for nozzle flow simulation. At the inlet of nozzle, reservoir stagnation conditions are employed. Pressure farfield condition is used for the outer boundaries. At a very high Reynolds number, the boundary layer developing at the nozzle wall is very thin, and its influence on the mean core of the jet is very weak.

W-1E-3. DETAILED ANALYSIS OF THRUST PLUME AND SATELLITE BASE REGION INTERACTION

J. G. KIM, *KAIST, Korea*, O. J. KWON, *KAIST, Korea*, M. J. YU, *KARI, Korea*, K. H. LEE, *KARI, Korea* and S. K. KIM, *KARI, Korea,* The interaction between thrust plume and satellite base region for the Korea multipurpose satellite-Ⅲ configuration was investigated by using direct simulate Monte Carlo calculations. For the accurate simulation of H2 collisions and rotation-translation transition, a variable soft-sphere model and a recent rotational relaxation model of N2 and H2 were used. For the investigation of the interaction between thrust plume and base region, the number density distribution for each species, translational and rotational temperature distributions, heat flux, and pressure were examined by DSMC calculations. It was found that most of the surface properties are affected by H2 collisions and a strong non-equilibrium state is observed on the S-band antenna and at the base region. It was demonstrated that an accurate model is needed to simulate H2 collisions and the rotation-translation transition. The results by the present calculation are more accurate than previous DSMC calculations because more accurate models were used in simulating elastic and inelastic collisions in the present study.

W-1E-4. EXPERIMENTAL INVESTIGATION ON HYSTERESIS PHENOMENA OF MACH DISK IN UNDER-EXPANDED AXISYMMETRIC JET

S. MATSUO, *Saga University, Japan*, T. HIGASHI, *Saga University, Japan*, T. SETOGUCHI, *Saga University, Japan*, H. D. KIM, *Andong National University, Korea,* When the high-pressure gas is exhausted to atmosphere from the nozzle exit, the under-expanded supersonic jet with the Mach disk is formed at a specific condition. The jet structure has been known as a fundamental phenomenon of the supersonic fluid mechanics. The jet is very important for some industrial devices and there exists many papers. These papers are on the steady jet as the ratio of reservoir stagnation pressure to back pressure is fixed. In two-dimensional under-expanded supersonic jet, it is reported that the hysteresis phenomenon for the reflection type of shock wave in the jet is occurred under the quasi-steady flow in the recent studies and the transitional pressure ratio between regular reflection and Mach reflection in the jet is affected by this phenomenon. The purpose of this study is to clarify the hysteresis phenomena for the reflection type of shock wave at the under-expanded axi-symmetric jet experimentally and to discuss the relationship between hysteresis phenomenon and rate of the change of pressure ratio with time. Furthermore, the effect of Mach number at the nozzle exit on hysteresis loop was investigated for two kinds of nozzle. The design Mach numbers of Nozzles A and B are M_e= 1.0 and 1.5, respectively. The Mach disk diameter and the distance from nozzle exit to Mach disk were obtained from shadowgraph pictures. As a result, in the under-expanded axi-symmetric jet, hysteresis phenomena for the configuration of shock wave were investigated experimentally. The phenomena were confirmed in the supersonic axi-symmetric jet at the present experimental condition. Furthermore, the range of hysteresis loop extended with an increase of Mach number at the nozzle exit and this phenomenon had the same characteristics as that in the transition between regular and Mach reflection of shock waves in two-dimensional flow field.

09:00 ~ 10:20 (Room106)

General Fluid Dynamics

Session Chair : Prof. M. Alam JNCASR/India

W-1F-1. THERMOMAGNETIC CONVECTION IN A MAGNETIC NANOFLUID LAYER

I. S. SHIVAKUMARA, *UGC-Centre for Advanced Studies in Fluid Mechanics, Department of Mathematics, Bangalore University, India,* Magnetic nanofluids are commercially manufactured stable colloidal liquids formed by suspending fine magnetic monodomain nanoparticles in non-conducting carrier liquids like water, heptane, kerosene, etc. Thermal convection in magnetic nanofluids has become a topic of current technical importance because magnetic forces can be used to create circulation of coolant in small passages where natural convection is either absent or ineffective. In the present paper, the linear stability of an initially quiescent magnetic nanofluid layer under the influence of an external magnetic field is investigated. The lower rigid-ferromagnetic boundary is heated by a constant heat flux, while at the upper rigid-ferromagnetic boundary a general thermal condition which encompasses fixed temperature and constant heat flux as particular cases is invoked. The resulting eigenvalue problem is solved numerically using the Galerkin technique and it is observed that the instability is always onsets into steady convection. Realizing this fact, closed form analytical solutions are also obtained using regular perturbation technique for constant heat flux thermal boundary conditions with wave number a as a perturbation parameter. It is found that the results obtained from these two techniques are in excellent agreement suggesting the analytical solution obtained for the constant heat flux thermal conditions is exact. It is also found that an increase in the magnetic number M_1, and the nonlinearity of fluid magnetization M_3, as well as decrease in the Biot number Bi, is to hasten the onset of thermomagnetic convection. To the contrary, the nonlinearity of fluid magnetization is found to have no effect on the threshold values for the onset of thermomagnetic convective instability in the case of constant heat flux thermal boundary conditions. Besides, comparison of results with earlier work is also made under the limiting case and good agreement is found.

W-1F-2. FRICTION FACTOR IN A V-SHAPED OPEN CHANNEL

Mirali MOHAMMADI, *Department of Civil Engineering,Urmia University, Urmia,* Alireza MOGHADDAMNIA, *Faculty of Natural Resources, University of Zabol, Iran,* The experimental study on the distribution of friction factor, f, of a V-shaped channel is examined as it occurs in sewers and culverts. Several series of experiments were conducted for measuring velocity and boundary shear stress. It may be seen that the Darcy-Weisbach friction factor, f, is more sensitive than the other resistance coefficients such as the Manning n. The contour plots of 2D isovels shows that the isovels are parallel to the channel boundary in a region close to the bed, and almost