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