W-1B-3. EULERIAN MODELING OF PARTICLE DEPOSITION IN A RESPIRATORY CYCLE USING HORSFIELD AND WEIBEL MODELS FOR WHOLE LUNG AND THE FIVE LOBES

S. S. KHALAFVAND, *Isfahan University of Technology, Iran*, M. S. SAIDI, *Isfahan University of Technology, Iran,* In this work, the dynamic deposition of particles in the lung is modeled based on the Eulerian-Eulerian approach. The whole lung is simulated and the deposition rate in each generation is derived by solving the aerosol General Dynamic Equation (GDE). All deposition mechanisms are considered and the effect of each mechanism for each particle size is studied. The one-dimensional GDE is solved by the fractional step method to obtain the size distribution of the inhaled particles in the lung. Also the alveolar region is considered to have expansion and contraction during a breathing cycle. In this article the growth and coagulation of particles are also modeled. In order to solve the GDE a computational method is implemented based on time-step splitting and subcycling approach, combined with a moving grid method for growth process. Comparison of the results with current experimental and numerical results shows a good agreement. Also it shows that larger particles deposit more on the upper region, while the smaller ones deposit more on the lower region of the lung. Including the effect of particle growth and coagulation increases the rate of deposition of smaller particles. The results show the maximum deposition occurs in right lower lobe and minimum deposition occurs in right middle lobe and that deposition fraction in Horsfield's model is lower than Weibel's model.

W-1B-4. AIR FLOW FIELD AND MICRO-PARTICLE DEPOSITION IN THE THREE GENERATION OF HUMAN LUNG

M. YOUSEFI, *Department of Mechanical Engineering, Isfahan University of Technology, Iran,* M. S. SAIDI, *Department of Mechanical Engineering, Sharif University of Technology, Iran,* The effect of flow structure and particle size on the transport and deposition patterns in a rigid, smooth-walled model of the human lung airways extending from trachea to the segmental bronchi is studied. In this work, based on the Horsfield (1971) morphometrical data of human lung, an out-of-plane model of three generation of human lung is produced. Particle deposition sites and efficiency studied for particles in the Stokes number range of, $0.025 \le Stk_{Trachea} \le 0.102$, at inspiratory flow rates of 30 lit/min, measured at trachea. A commercially available CFD code is used for numerical simulation. A lagrangian approach is employed, and one-way coupling is used between the continuum and dispersed phase, which allows particle tracking to be run as a post-processing calculation. The 3-D steady laminar flow is numerically simulated and then the particles trajectories are determined by numerically solving the Newton law. It is shown that the particle size has substantial influence on deposition, regarding both efficiency and location. Also it is noticed that for micron size particles deposition mainly occurs by inertial impaction at upper airways. Deposition efficiency with parabolic velocity at the inlet is about 50% more than realistic inlet condition.

09:00-10:20 (Room103)

Flows Stability (I)

Session Chair : Prof. Y.-W. Lee, Pukyong Univ/Korea

W-1C-1. HYDROMAGNETIC INSTABILITY OF GIESEKUS FLUIDS IN PLANE POISEUILLE FLOW

S. M. TAGHAVI, *University of Tehran, Iran,* K. SADEGHY, *University of Tehran, Iran,* The effect of transverse magnetic field is investigated on the instability of Giesekus fluids in plane Poiseuille flow for small magnetic Prandtl numbers. Our approach is a classical one in which stability of flow to small, two-dimensional disturbances will be studied using linearized theory. The system of equations so obtained will be solved using pseudo-spectral method to determine the effect of parameters such as magnetic number, Weissenberg number, mobility factor, and viscosity ratio on the least stable eigenmode. It is found that magnetic field has always a stabilizing effect on plane Poiseuille flow of Giesekus fluids. In contrast, solvent viscosity, fluid's elasticity, and mobility factor may have a stabilizing or destabilizing effect depending on their magnitude being smaller or larger than a critical value.

W-1C-2. THREE DIMENSIONAL VORTICAL STRUCTURES AROUND A LOW-ASPECT-RATIO WING AT LOW REYNOLDS NUMBER

Jongkook SEONG, *Seoul National University, Korea*, Hyungmin PARK, *Seoul National University, Korea*, Byungdo LEE, *Hynix Semiconductor Inc., Korea,* Haecheon CHOI, *Seoul National University, Korea,* Understanding three-dimensional flow structures around insect wings, operated at low Reynolds numbers (Re < 10^4) both in gliding and flapping flights, is important in developing the micro-air vehicles. Nevertheless, the detailed investigation on the three-dimensional flow phenomena has not been done carried out thoroughly. In nature, the butterfly is one of the representative flying insects having low-aspect ratio wings and performing both gliding and flapping flights effectively. Thus, the investigation of the flow around a butterfly in a gliding flight may provide an insight on the generation of three-dimensional vortical structures and their role in producing the drag and lift forces. In our previous study, we performed an experiment on a swallowtail butterfly in gliding flight at the Reynolds number of 14,400. We observed the variation of leading-edge and wing-tip vortices with the attack angle. Also, we found that the hind-wing tails of the swallowtail butterfly improve the aerodynamic performance of gliding flight such as the increase in the lift-to-drag ratio and longitudinal static stability. In the present study, we perform numerical simulations of flows around a gliding swallowtail butterfly and an inverse delta wing, respectively, having low-aspect-ratio wings at low Reynolds number (Re=1,000), to investigate three-dimensional vortical structures around the wings. The angle of attack considered ranges from 5° to 30°. Four vortical structures are identified: leading-edge, trailing-edge, wing-tip and hairpin vortices. Due to the thin and sharp leading edge of the wing, the flow separates at the leading edge at small attack angle ($\alpha = 10°$) and massive separation occurs at $\alpha > 15°$. Wing-tip vortices are generated at $\alpha > 10°$. The drag and lift forces are significantly affected by those vortical structures. In addition to the well-known leading-edge and wing-tip vortices, the hairpin vortex is for the first time observed. The generation of hairpin vortex induces relatively low pressure region on the upper wing surface, resulting in the increase in the lift force on the wing.

W-1C-3. SPATIAL STABILITY OF COMPRESSIBLE PLANE COUETTE FLOW

M. MALIK, *Department of Aerospace Engineering, Indian Institute of Science, Bangalore, India,* Meheboob ALAM, *Engineering Mechanics Unit, JNCASR, Bangalore, India*, J. DEY, *Department of Aerospace Engineering, Indian Institute of Science, Bangalore, India,* Spatial stability of compressible plane Couette flow is studied. The spatial spectrum of the streamwise wavenumbers in the complex plane resembles that of incompressible case. The instability is caused by a mode which is second least-decaying at very low frequencies, and which becomes most unstable as the frequency is increased. The growth-rate contours in parameter space suggest that the spanwise modulation stabilizes the flow for the parameter ranges studied. An analysis of the energy contained in the least-decaying mode reveals that the instability is due to the work by the pressure fluctuations and an increased transfer of energy from the mean-flow. In the case of oblique modes the stability at higher spanwise wavenumber is due to higher thermal diffusion rate. At high frequencies there are disjoint regions of instability at chosen Reynolds number and Mach number. The stability characteristics in the inviscid limit is also presented. The increase in Mach number and frequency is found to destabilize the inviscid unstable modes. The inviscid mode, having phase-speed within the range of mean-flow velocity, is found to be having non-zero growth/decay-rate in agreement with the nature of absence of generalized inflectional point in the mean-flow. The behaviors of the non-inflectional neutral modes are similar to that of compressible boundary layer. A leading order viscous correction reveals that the neutral and unstable modes are destabilized by the no-slip enforced by viscosity. The viscosity has a dual role on the stable inviscid mode.

W-1C-4. EFFECT OF DIAPHRAGM RUPTURE PROCESS ON S HOCK TUBE FLOWS WITH NON-EQUILIBRIUM CONDENSAT ION IN RAREFACTION WAVE

S. MATSUO, *Saga University, Japan*, H. NISHI, *Saga University, Japan,* T. SETOGUCHI, *Saga University, Japan*, H. D. KIM, *Andong National University, Korea,* For a shock tube with diaphragm, in order to simul ate the flow induced simultaneously with diaphragm rupture accurately, it is important to estimate the process of diaphragm rupture. This bec omes the important point of caution for investigation of formation dis tance of shock wave and non-equilibrium condensation in the unstead y expansion wave. In the flow in shock tube, there are some research es for the effect of rupture time of metal diaphragm and its opening time on shock Mach number. From these researches, it is found that diaphragm rupture process is not instantaneous and the process is thre 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