BASED TURBULENCE MODELS FOR ATTACHED AND SEPARATED FLOWS

D. S. KULKARNI, B. N. RAJANI and S. MAJUMDAR, *CTFD Division, NAL(CSIR), Bangalore, India,* B. RAGHAVENDRA, *Department of Mechanical Engineering, Sir MVIT, Bangalore, India,* RANS (Reynolds Averaged Navier-Stokes) solvers coupled to appropriate eddy viscosity based turbulence models have emerged as the most cost-effective approach today for CFD (Computational Fluid Dynamics) analysis of industrial turbulent flow problems. Unfortunately there is no single reliable turbulence model which is universally applicable for any flow whatsoever. A common disadvantage of most of the widely used two-equation turbulence models is the inaccurate prediction of the near wall flow behaviour where the normal stress anisotropy arising out of the physical blocking of the wall-normal fluctuation cannot be captured by the simplified scalar eddy viscosity approach, even using the exponential damping functions in the near wall zones. The other important mean flow condition for which the popular two-equation turbulence models are often observed to be inadequate and inaccurate is the adverse pressure gradient and eventually flow separation. While considering the performance of a turbulence model, the main challenge is how accurately the model can capture flow separation in wall bounded turbulent flows– the inception of separation, reattachment and post separation recovery of the flow field.   The present study is aimed at calculation of three different wall bounded two-dimensional turbulent flow situations, viz., fully developed flow through a plane channel, flow through a plane asymmetric diffuser and flow past an airfoil   by solving Reynolds Averaged Navier Stokes (RANS) equations coupled to different eddy-viscosity based turbulence models. The turbulence models tested in the present study are low Reynolds number version of the k-ε model of Chien , k-ω model of Wilcox, Shear Stress Transport model of Menter as a weighted combination of k-ε and k-ω model and the simplified second moment closure based $k - \varepsilon - \overline{v^2} - f$ (so-called V2F model) model of Durbin. The relative performance of these different turbulence models is assessed through comparison of the computation results to corresponding DNS and/or measurement data, available in unclassified literature.

W-3C-4. NUMERICAL SIMULATION OF IONIZED PARTICLE MOTION IN FLUID FLOW

Y. KOJIMA, Y. ENDO, K. SAKABE, H. ISHIKAWA, *Tokyo University of Science*, *Japan*, T. SETO, *Kanazawa University*, *Japan,* For the purpose of contribution to the reduction technology of nanometer-scale chemical pollutants, DIMA was developed to measure and classify such nanometer-scale particles.   DIMA has measuring principles based on differential mobility analyzer (DMA), with various improvements given to measure the nanometer-scale particles and ion clusters with high resolution and high sensitivity.   Especially, DIMA made possible to measure the ion cluster. The DIMA analyzing region has a coaxial cylinder structure composed of an outer housing and an inner center rod.   Nanometer-scale particles are ionized by electric charge, introducing into the analyzing region in inside of the DIMA with a carrier gas.   In the analyzing region, ionized particles are forced to flow downward with clean sheath gas.   The inner center rod is charged with high voltage and the outer housing is grounded, creating an electric field to attract ionized particles to drift toward the center rod.   The particle path in the analyzing region depends on ion-mobility, and by giving a certain voltage of the center rod, it allows specific particle to correct from a sampling slit on the surface of the center rod. However, measurement error sometimes occurs by the disorder element which inheres in the internal flow of fluid drag, turbulence and the influence of Brownian motion. Therefore, it is important to explain ionized particle motion in flow field by numerical simulation.   This paper proposes a three-dimensional numerical model to explain such motion inside the DIMA, with electric field, fluid drag, turbulence, Brownian motion, and repulsion taken into consideration. Three dimensional computational domain where is a portion of one fifth of DIMA coaxial analyzing region for the circumferential direction was used in this study.   Ionized particle are introduced from an inlet slit which is located at the upstream outer housing wall.   The width of inlet slit is denoted by "W".   The bottom wall represents the surface of inner center rod which is charged with high voltage.   An outlet slit, at which is located downstream of 100 times the width of the inlet slit, is used to capture ionized particle. In this study, ionized particle motion was calculated by overlapping the effect of flow field and the electric field. First, flow field was determined by solving the governing equations of the Navier-Stokes and the continuity equation under the assumption that the internal flow of the DIMA to be incompressible and laminar.   The one-way coupling method was applied to disregard the effect of ionized particle motion on flow.   The Reynolds number based on the flow velocity and inlet width of the sheath gas and the kinematic viscosity is in the range of 3,000 to 18,000.   The ion cluster model was not taken in consideration in

this study. From the result of the comparison of repulsive force with electric field, it is found that the repulsive force enhanced the dispersion of ionized particle motion.   As for the dispersion of captured ionized particles, the effect on differences of ionized particles motion becomes dominant in order of repulsion, Brownian motion, turbulence and electric field.

13:20 ~14:40 (Room104)

Aerodynamics (V)
Session Chair : Prof. M. Takao, Matsue National College of Tech/Japan

W-3D-1. COMPUTATION OF FLOW FIELD AROUND RE-ENTRY CAPSULE AT SUPERSONIC SPEEDS

S. DAS, *Department of Space Engineering & Rocketry, B.I.T Mesra - Ranchi, India,* J. K. PRASAD, *Department of Space Engineering & Rocketry, B.I.T Mesra - Ranchi, India,* Re-entry missions are being thought upon as alternative to conventional launch vehicles and winged body due to various advantages. A reentry capsule will have the capability of vertical landing with advantages of more space for experimental payloads / instruments, brought back to earth, after completing the space missions. The blunt configurations are adopted to decelerate the vehicles for safer missions of return to earth. Reentry capsules generally exhibit aerodynamic instabilities due to formation of wake and possible flow separation at low altitudes, which may be detrimental for the mission. In general, flow field around a typical blunt capsule at supersonic speeds are characterised by presence of a strong detached bow shock, expansion at the shoulder, and a separated boundary layer at the rear surface. The separation depends on freestream Mach number, angle of incidence, Reynolds number etc. This shows the presence of complex flow structure around capsule with different flight conditions. The characterisation of flow field in the wake of the capsule is also important with a view of aerodynamic stability of the capsule. Numerical simulations have been made to obtain the flow field over a blunt body reentry capsule at supersonic speeds. Experiments involving schlieren and measurement of static pressures were made at a Mach number of 2. Comparison indicates good agreement between experiments and the computation. The flow around the reentry capsule has been computed at Mach numbers of 2, 3 and 5. Existence of a complex flow in the wake is observed which is characterized by re-circulating flow, separation zone, re-compression, vortex core and stagnation points. The change in wake flow field at lower Mach number is more predominant in comparison to higher Mach numbers which might be responsible to affect the overall aerodynamic parameter of the capsule.

W-3D-2. THE INFLUENCE OF REYNOLDS NUMBER ON THE AERODYNAMIC CHARACTERISTICS OF AIRFOILS WITH TRIANGULAR SHAPED THROUGH DAMAGE

Bahareh YAHYAVI, *K.N.Toosi University of Technology, Iran*, Masoud MIRZAEI, *K.N.Toosi University of Technology, Iran*, Mahmoud MANI, *Board Member, Center of Excellence in Computational Aerospace Engineering, Amirkabir University of Technology, Iran,* In this paper flow around a full span NACA 64_1-412 airfoil with two different orientation of triangle shaped damage was experimentally investigated using balance measurements and flow visualization. To assess the influence of Reynolds number on results, models of airfoil with 150mm and 100mm chord were used. Our studies were carried out   in a low turbulence, closed circuit, wind tunnel at Aerospace engineering department of AmirKabir University of technology, with test section of 0.45×0.45 m. The experiments were done at air speed of   35 m/s. Each model was tested in both undamaged and damaged states. The standard tuft technique was used for flow visualizations. At any incidence, the increments are defined as: $dC_l = (C_l)_{damaged} - (C_l)_{undamaged}$, $dC_d = (C_d)_{damaged} - (C_d)_{undamaged}$, $dC_m = (C_m)_{damaged} - (C_m)_{undamaged}$ For damaged model, increasing incidence generally resulted in greater lift loss, higher drag and more negative pitching moment. Inverse triangle, in comparison with the right triangle case, resulted in the smaller drag increments and reduced lift increments. For the pitching moment increments the inverse triangle results are more negative than for the right triangle. Increasing incidence resulted in a stronger jet, with more significant effects on the flow over the airfoil. Changing the orientation of the damage to produce the inverse triangle resulted in significant changes in the flow characteristics. The main feature of these flows is identical to those previously described by Irwin for circular damage and also Mani & Render for triangular damage [3]. Our results indicated that the coefficient changes were dependent to the Reynolds number over the range tested, and by varying Reynolds number, the trends were largely changed. At lower Reynolds number, the rate of d[C_l] change increased with incidence dramatically, and the rate of d[C_d] change decreased over the full incidence

range.

W-3D-3. DESIGN AND TESTS OF A NATURAL LAMINAR FLOW AIRFOIL
Yung-Gyo LEE, Cheolwan KIM, Kijung KWON, Tae-Hwan CHO, Jae-Yeul SHIM, Eung-Tae KIM and Dae Sung LEE, *Korea Aerospace Research Institute, Daejeon, Korea,* Drag reduction is one of main concerns for commercial aircraft companies than ever because fuel price has been tripled in ten years. In this research, Natural Laminar Flow airfoil is designed and tested to reduce drag at cruise condition, c_l=0.3, Re=3.4x10^6 and M=0.6. NLF airfoil is characterized by delayed transition from laminar to turbulent flow, which comes from maintaining favorable pressure gradient to downstream. Transition is predicted by solving Boundary Layer equations in viscous boundary layer and by solving Euler Equation outside the boundary layer. Once boundary layer thickness and momentum thickness are obtained, e^N-method is used for transition point prediction. Empirically adjusted e^N-method is known to be accurate and fast for transition prediction. A new NLF airfoil, KARIFOIL, with maximum thickness of 15% is designed for a very light jet. It is modified from NLF-0115 airfoil by reducing leading edge radius and by shifting maximum thickness position downstream. Camber is also increased to shift up the drag bucket for better performance at climb conditions, c_l=0.6. Transition position and drag polar are computed and compared for various Reynolds numbers. Transition is delayed at upper surface comparing to NLF-0115, which results in less drag than NLF-0115. It is interesting to notice that drag of KARIFOIL is higher than that of NLF-0115 at low Reynolds numbers, but drag bucket gets wider and drag is reduced as Reynolds number increases. Eventually, the designed airfoil has less drag at a cruise condition as well as a climb condition than drag of NLF-0115 at cruise condition, Re=3.36x10^6. Pitching moment shows more negative values. It is important to notice that transition at high angle of attack locates almost leading edge to minimize aerodynamic change by contaminations like bugs or rain drops during take-off and landing. KARIFOIL is tested in KARI 1-m Low speed wind tunnel to investigate subsonic characteristics. M$\fallingdotseq$0.1 and Reynolds number ranges from 3x10^5 to 9x10^5. At low Reynolds numbers (< 9x10^5), test results show good agreements with prediction. Drag bucket is shown clearly. And it is wider and minimum drag is smaller for larger Reynolds numbers although Reynolds numbers are lower than the design target. High subsonic and transonic characteristics of KARIFOIL are examined in a transonic wind tunnel facility. Mach number ranges from 0.5 to 0.7 and Reynolds number from 3.3x10^6 to 4.3x10^6. Unfortunately it seems that flow over airfoil is turbulent by free stream turbulence and vibration of a model. So, minimum drag is much higher than conventional NLF airfoils and data can't show clear tendency for Reynolds number variation. For turbulent flow, tests results show the same tendency as prediction. Flow visualization doesn't show any evidence of transition. As results, KARI's NLF airfoil is designed and shows better characteristics than NLF-0115. The characteristics are tested and verified at low Reynolds numbers, but at high Reynolds numbers, laminar flow characteristics are not obtainable because of fully turbulent flow over airfoil surfaces. Precious experiences is, however, relating NLF airfoil design, subsonic and transonic tests are acquired.

W-3D-4. AERODYNAMIC INVESTIGATION OF FLOW THROUGH A CENTRIFUGAL COMPRESSOR STAGE
R. RAJENDRAN, *NAL, India*, S. RAMAMURTHY, *NAL, India*, P.MOHANAN, *NITK, India,* In a centrifugal compressor the work is imparted on the impeller to get higher total pressure of the working fluid. A diffuser is employed at the down stream of the impeller for the conversion of kinetic energy of the flow coming out of the impeller into static pressure. The overall efficiency of the compressor is dependent on the design of both impeller and diffuser. The vane diffuser reduces the operating range, however by proper setting of the diffuser with reference to impeller; it is possible to achieve good stage performance. The setting angle of the diffuser with reference to the impeller plays a crucial role on the stage performance. This paper was aimed to experimentally investigate the flow behavior in a centrifugal compressor stage with detailed flow inside three different setting angles of the vane diffusers. The experiments were carried out at different speeds ranging from 15000 to 20000 rpm in a closed circuit centrifugal compressor test rig. Static pressure measurements were carried out at impeller shroud from inlet to exit of the impeller to study the flow behavior in the impeller at off design conditions. Static pressure measurements on the suction and pressure surface of the vane diffuser and also on the diffuser channel from diffuser leading edge to diffuser exit at different radius to study the effect of pressure recovery in diffuser on the overall stage performance. Similarly unsteady flow measurements were carried out at impeller shroud and diffuser channel using miniature high response Kulite transducers to study

the unsteady flow behavior at the compressor stage with three different configurations of the diffuser. From the experimental results an optimum vane configuration is selected to achieve good stage performance.

13:20 ~ 14:40 (Room105)
Geophysical Fluid Dynamics (II)
Session Chair : Prof. N. Huang, Lanzhou Univ/China

W-3E-1. NUMERICAL STUDY ON INFLUENCE OF RAINFALL INTENSITY ON HILLSLOPE EROSION
Y. AN, *IMECH CAS, China*, Q. Q. LIU, *IMECH CAS, China,* Rainfall intensity is one of the most important parameters of influencing soil erosion on hillslopes. The phenomena, that erosion amount rises with the increase of rain intensity under the same precipitation, is widely observed. It is commonly accepted that rill erosion plays an important role in this complex process. Different rain intensity would induce different rill condition which in turn results different erosion amount. In order to consider the impact of various rill conditions and distinguish the contribution of rill and interrill area, a two-dimensional erosion model, which includes a one-dimensional rill component and a two-dimensional interrill component, is proposed. This model calculates rill and interrill processes respectively, which indicates a more reliable rill flow character can be obtained. Several sets of experimental data are used to verify the model and a good agreement is observed. The effect of rain intensity is discussed by considering influences of rill development stages which directly corresponds with rain intensity. Influences of parameters which represent rill development stages are discussed here. The hillslope with approximately parallel rills is generalized to a numerical plot containing one rill and its interrill catchment. Basing on this platform, a series of numerical experiments discussing the mentioned parameters, are carried on. From the results of numerical experiments, the following preliminary results can be obtained: (1) A slope with rill might produce more erosion than a slope without rill; (2) Along with the increase of rain intensity, a peak value of erosion capability of a single rill appears.; (3) The increase of rill intensity, which includes increase of rill number and rill length, would contribute to the augmentation of hillslope erosion. More assured conclusion on how rainfall intensity effecting erosion amount might be obtained by simulating the development process of rill on hillslopes.

W-3E-2. NUMERICAL PREDICTIONS ON ATMOSPHERIC DISPERSION OF POLLEN IN EASTERN AICHI, JAPAN
N. SEKISHITA, *Toyohashi Univ. of Tech., Japan*, H. MAKITA, *Toyohashi Univ. of Tech., Japan,* A computer simulation was conducted for the prediction of pollen dispersed in atmospheric turbulence in Aichi, Japan. Recently, hay fever becomes serious problems in Japan due to atmospheric diffusion of pollen. This pollen dispersion is affected by time when trees release pollen, velocity fields, weather condition, etc. The precisely prediction of atmospheric dispersion of pollen can help many people who are sick with hay fever. The diffusion equation was numerically solved based on velocity data calculated by mesoscale weather model, MM5. The present prediction was carried out on March 6th, 2007 when a lot of pollen measured in the eastern Aichi by Ministry of the Environment, Japan. Three velocity components, u, v, w were gotten at each 2km on north-south and east-west lines in the simulation area: 136.8–137.9 of east longitude and 34.5–35.3 of north latitude. The falling velocity of the pollen in the atmospheric turbulence was assumed to be the terminal velocity of the pollen and its value was 0.036m/s in this case. The quantity of dispersing pollen from trees was assumed to be constant along a forest in this simulation; pollen density c was always 100 pieces/m^2 in the forest distributed in the present calculation area. The pollen density settled down on the ground was calculated from 14:00 to 14:30 on March 6th, 2007. Since comparatively strong north-east winds blew in this period, we observed the high distribution of the pollen density in south-east side of the source (forest). Velocity vectors and the contour map of pollen density on a horizontal cross section were also estimated. The pollen source was existed on the windward slope of a small mountain. This result showed that the pollen dispersed leeward of the mountain due to wind. The pollen dispersion in the atmosphere was simulated successfully. In the near feature, the present simulation will improve by precisely modeling release time from trees, the falling velocity of pollen, the effects of geographical features, etc. And, the information of pollen dispersion will be provided for local people.

W-3E-3. NUMERICAL ANALYSIS ON IMPROVEMENT EFFECT OF WIND SHEAR BY A STRUCTURE INSTALLED UPSTREAM OF A WIND TURBINE