

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 MVTI, 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 $k-\epsilon-\overline{v^2}-f$ (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 641-412 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: $dC_l = (C_l)_{\text{damaged}} - (C_l)_{\text{undamaged}}$, $dC_d = (C_d)_{\text{damaged}} - (C_d)_{\text{undamaged}}$, $dC_m = (C_m)_{\text{damaged}} - (C_m)_{\text{undamaged}}$. 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