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≒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, 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

H. G. KIM, *KIER (Korea Institute of Energy Research), Korea*, S. W. WOO, *KIER, Korea,* This paper presents the improvement effect of wind shear by a structure installed upstream of a horizontal axis wind turbine. The atmospheric boundary layer has low wind speed near the ground surface due to friction but wind speed increases according to height above ground which leads a vertical wind speed profile having wind gradient, i.e. wind shear. Although such vertical wind speed profile gets strong wind loads while the wind turbine blades passes through the upper semicircle of the hub, there is a danger of decreasing the durability of wind turbine due to periodic aerodynamic fatigue loads as it gets relatively weak wind loads at the lower semicircle of the hub. Also, it can be the cause of vibration and noise due to eccentricity. Therefore, this study has performed a research on the method of uniformly distributing the wind speed delivered to the wind turbine blades by improving the wind shear. As an analysis method, the two-dimensional computational flow analysis has been carried out by assuming the structure installed upstream of a wind turbine as sold fence or windbreak forest. As a result, both the structure assumed as solid fence and the one assumed as windbreak forest showed near-uniform wind speed profile than in case of not installing the structure and the mean wind speed was shown as the one that can be increased. Assuming a 1.5MW wind turbine for reference, the difference between the wind speed at the top of blades and the wind speed of at the bottom of blades was shown as 9.5% in case of using solid fence and 12.5% in case of using wind break forest so that the wind speed was considerably reduced from 18.9% when nothing has been installed. That is because such a ground structure introduces a recirculation zone at the back of structure and this recirculation zone acts as a virtual streamline shaped hill to increase wind speed just like the hill effect with simple structure. Also in case of windbreak forest showing porous nature, it has the similar effect of increasing the wind speed by creating recirculation zone. But in case of the windbreak forest having porosity, it could be confirmed that the length of recirculation zone becomes shorter and this has verified the fact that a uniform wind speed can be delivered and also can expect the wind speed increasing effect if the structure is installed by adjusting porosity according to the surrounding environment and layout. Finally, the designing conditions on the installation position between a wind turbine and the structure is being presented by non-dimensional wind speed profiles.

W-3E-4. SOME INVESTIGATIONS INTO NEAR SURFACE WIND AND SALTATION INTENSITY IN MINQIN AREA

F. SHI, N. HUANG, X. J. ZHENG, *Key Laboratory of Mechanics on Western Disaster and Environment, Lanzhou University, China,* In recent decades, the research on the windblown sand movement has been intensified in order to meet the need of preventing from the increasing endangerment of wind erosion, desertification and dust storm. Current research on sand saltation concentrates on wind tunnel experiment, theoretical analysis and numerical simulation of sand saltation at ideal and controllable conditions and most field observation results are hour or day length averaged sand flux and wind velocity. Whereas the results on turbulent characteristics of near surface wind in real atmospheric boundary layer and the effects of fluctuations in wind velocity on sand saltation appear to be much fewer although these topics have obtained increasing recognition of importance in recent decades. Based on some instruments imported from abroad, we jointly developed a system with colleagues from The Wind Erosion and Water conservation Research Unit of US Agriculture Department, which consists of four lightweight fast-responding cup anemometers, four SENSITs, eight BSNE collectors, the HMP50 Temperature and Relative Humidity Probe and can simultaneously measure pulsed wind velocity at near surface, sand transport intensity, sand mass flux, temperature and humidity are measured with 1Hz frequency on sand dune in Minqin area, which locates between edges of the Badain Jaran Desert and the Tengger Desert. The analysis on the measured data shows that wind fluctuation with high frequency and sand saltation is with high level of intermittency accordingly. The moving sands can fully respond to the fluctuation of wind whose frequency is lower than 1/30 Hz. And, only when the mean time step is more than 30s, can the average wind velocity distribution obey logarithmic distribution.

13:20 ~14:40 (Room106)

Convection and Buoyancy – Driven Flows (I)

Session Chair : Prof. K. Mansour, Amirkabir Univ of Tech/Iran

W-3F-1. EFFECT OF EVAPORATOR SURFACE ROUGHNESS ON THE PERFORMANCE OF A TWO-PHASE CLOSED LOOP THERMOSYPHON

M. A. R. AKHANDA, *IUT, Bangladesh,* S. L. MAHMOOD, *IUT, Bangladesh,* A. B. M. N. BAGHA, IUT, *Bangladesh,* A two-phase closed loop thermosyphon has been designed, fabricated and tested. This thermosyphon consists of four components in its loop: an evaporator with boiling enhancement structure, vapor rising tube, condenser and condensate return tube. Tests are conducted at atmospheric pressure to assess the effects of evaporator surface geometry using three working fluids (acetone, ethanol and methanol). Evaporator surface is heated by using an electric capsule heater which is connected to the A.C power supply. Heat supply is varied using a voltage regulator which is measured by a precision ammeter and a voltmeter. Condenser section is cooled by natural circulation of air. Temperatures at different locations of the evaporator surface are measured using calibrated K type thermocouples. Four different evaporator surfaces namely smooth surface (SS), semicircular ribbed surface (SCRS), triangular ribbed surface (TRS) and saw tooth ribbed surface (STRS) have been tested in this study. It is found that, for ethanol, at 20°C wall superheat, heat flux dissipated from saw tooth ribbed surface (STRS) is around 35% higher, from semicircular ribbed surface (SCRS) is around 25% higher and from triangular ribbed surface (TRS) is around 16% higher than that of smooth surface (SS) respectively. Among the working fluids used, heat flux dissipation from evaporator surface using ethanol is about 1.3 times higher than that of methanol and about 2 times higher than of that of acetone respectively. Thus saw tooth ribbed surface (STRS) shows the best performance among all the evaporator surfaces tested in this study and among all working fluids used ethanol's performance is the best.

W-3F-2. INCOMPRESSIBLE MULTI-RELAXATION-TIME LBM WITH NON-UNIFORM MESH FOR LES OF RAYLEIGH-BÉNARD CONVECTION FLOW

A. R. RAHMATI, *Isfahan University of Technology, Iran*, M. ASHRAFIZAADEH, *Isfahan University of Technology, Iran*, E. SHIRANI, *Isfahan University of Technology, Iran,* Rayleigh–Bénard convection flow is an original phenomenon which occurs in a wide variety of atmospheric and industrial applications. Various numerical schemes have been implemented to study this problem, including the lattice Boltzmann method (LBM) which has appeared as one of the strongest CFD methods for simulating fluid flows and modeling physics in fluids in recent years. In the present work, the application of incompressible MRT-LB model for large-eddy simulation (LES) of turbulent thermally driven flows is considered. A Taylor series expansion and least square based Lattice Boltzamnn method (TLLBM) has been implemented in order to use a non-uniform mesh. It permits to reduce the required mesh size and consequently the computational cost to simulate the turbulent buoyant flow fields. The implementation is discussed in the context of a D2Q9-MRT-LB model in conjunction with the Smagorinsky subgrid closure model. The MRT-LB-LES is applied to a two-dimensional turbulent Rayleigh-Bénard convection flow at different Rayleigh numbers for Prantdl number of 0.71. Results show that the calculated Nusselt number is over predicted in comparison with the Nusselt number computed by the empirical formula at higher Rayleigh numbers. The over-prediction of the Nusselt number may be due to the SGS model through the evaluation of the SGS heat flux or due to insufficient spatial resolution.

W-3F-3. EFFECT OF WALL-NORMAL FLOW ON HIGH Sc NATURAL CONVECTION BOUNDARY LAYERS

G. Vijaya Rama REDDY and Baburaj A. PUTHENVEETTIL, *Department of Applied Mechanics, IIT Madras, India,* We investigate the effects of wall-normal velocity (V_b) on high Schmidt number (Sc>600) natural convection boundary layers formed on permeable horizontal surfaces. Using integral boundary layer equations, we define a blowing parameter, $S = (Re_L^5/Gr_L)^{1/8}$ to characterize the strength of blowing relative to the buoyancy and the viscous effects. The analysis is performed for $0.1 \le S \le 0.26$. The upper limit being given by $Re_\delta = V_b\delta_v/\nu < 1$ so that inertial effects in the boundary layer are small. The lower limit of S ensures that there is negligible diffusive mass transfer in the species boundary layer. As expected, blowing increases the velocity boundary layer thickness and the species boundary layer thickness; the effect is felt more on the species boundary layer thickness. Blowing also increases the horizontal velocity in the boundary layers. We show that the species boundary layer thickness scales as $x(Re_x/Gr_x)^{1/4}$ while the horizontal velocity scales as $V_b\,(Gr_x/Re_x)^{1/4}$.

W-3F-4. EFFECT OF INCLINATION ANGLE ON MIXED CONVECTION IN A LID-DRIVEN ENCLOSURE WITH INTERNAL HEAT GENERATION

A. K. M. SADRUL ISLAM, *IUT, Bangladesh*, G. SAHA, *AUST, Bangladesh*, S. SAHA, *BUET, Bangladesh,* M. Q. ISLAM, *BUET,*