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^{o}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,* *Bangladesh.,* The problem of steady, laminar and incompressible flow in a tilted lid-driven rectangular enclosure in the presence of internal heat generation is hereby investigated. Both the left and right vertical walls of the enclosure are the cold walls, the bottom wall is maintained at a constant heat flux that is moving in its own plane at a constant speed, while all other walls are fixed and top wall of the enclosure is adiabatic. Non-dimensional governing equations are solved by using combined finite element method. Numerical simulation of mixed convection inside a lid-driven inclined enclosure has been performed for inclination angle 0º, 10º, 20º and 30º for the present analysis. Every simulation is carried out for Richardson numbers 0 to 10, keeping Pr, Re and Δ equal to 7.0, 500 and 2 respectively. Flow and heat transfer characteristics via streamlines, isotherms, and average Nusselt number are presented. It is observed that both the inclination angle and Richardson number significantly influence the heat transfer process from a lid-driven enclosure having internal heat generation. In the flow patterns, a dominating unicellular structure is observed in all cases due to the shear force induced by the moving lid. But at low Richardson number, the cavity is filled with a major recirculating cell and two minor eddies. On the other hand, at higher Richardson number, the dominating buoyancy effect suppress one of the minor eddies located at the top-left corner. Significant increments in the average Nusselt number are observed with the increase of inclination angles. Moreover, a transition point of the Nusselt number is identified at Ri = 2 for the present mixed convection problem that differentiate the dominance between the forced and natural convection.

15:00 ~ 16:20 (Room103)

Turbulence Simulation

Session Chair : Prof. C. F. Li, Jiangsu Univ/China

W-4C-1. LARGE EDDY SIMULATION OF THE TURBULENT ROTATING CHANNEL FLOW

Z. X. YANG, G. X. CUI and C. X. XU, *Department of Engineering Mechanics, Tsinghua University, Beijing, P.R. China,* This paper investigates flow properties of a turbulent rotating channel flow by large eddy simulation (LES). The precise prediction of rotating turbulent flows is significant in geophysical science and turbo-machinery. Usually they are unsteady high Reynolds number flow and large eddy simulation is considered as a feasible numerical method for predicting such kind of flows. Rotating turbulence is anisotropic in both resolved and subgrid scale turbulence; hence a suitable subgrid model, which can take in to account the anisotropic transfer of turbulent energy between resolved and subgrid scale turbulence, is important for precise numerical prediction of rotating turbulence by LES. The authors proposed a new subgrid model for anisotropic turbulence by means of generalized Kolmogorv equation for resolved scale turbulence [Cui et al., 2007] and the model has been applied in numerical simulation of homogeneous rotating turbulence with considerable success while conventional models, such as Smagorinsky and spectral models, failed to predict the major properties of rotating turbulence. In this paper we use new model (Cui, et al. 2007) in prediction of turbulent rotating channel flow and the LES results. The numerical method is as follows. The spectral method employing Fourier series and Chebyshev polynomials is used for spatial discretization and the time splitting method with third-order accuracy is adopted to carry out time advancement. The Reynolds number is Re=2666 (based on half channel width and mean bulk velocity). The computational domain is $4\pi H \times 2H \times 2\pi H$ with $128 \times 128 \times 128$ grids in DNS while $32 \times 64 \times 32$ in LES. The LES computation is also completed with different subgrid models and compared to the DNS results. Most of results obtained by the new model are better than those by the dynamic model and Smogorinsky model. The results indicate that the new rational subgrid model with consideration of anisotropic effect is suitable for anisotropic turbulence with both shear and rotation.

W-4C-2. DEVELOPMENT OF A LARGE EDDY SIMULATION TECHNIQUE ON UNSTRUCTURED MESHES

J. Y. YOU, *KAIST, Korea* and Oh Joon KWON, *KAIST, Korea,* Until recently, flow around bluff bodies has been studied by numerous researchers through CFD and experimental investigations. For bluff bodies, the flow pattern in the wake is very complex and contains several different scales of fluctuating vortical structures, and thus simulation of the flow based on RANS(Reynolds-Averaged Navier-Stokes) approach has limitations to properly capture the detailed time variation of fluctuating eddies. Therefore, a more accurate numerical flow simulation technique is required. In the present study, an unstructured compressible LES(Large Eddy Simulation) technique based on the Smagorinsky model has been developed for the investigation of the flow around bluff bodies. For the numerical method, a cell-centered finite-volume scheme was adopted. The inviscid flux terms were discretized by using 2nd-order Roe's FDS, and the viscous fluxes were computed based on central differencing. To reduce the computational burden for resolving the boundary layer, a wall function approach was adopted. A Runge-Kutta four-step method was used for time integration. The geometry and the flow domain were modeled by using unstructured meshes. Large eddy simulations were conducted for a square cylinder and a sphere to validate the present method. To better capture the eddies, dense cells were distributed inside the wake region. The predicted streamwise velocity distribution in the wake region, the Strouhal number, drag coefficient, the vortical structures downstream of the bodies showed good agreements with experimental data and those of other researchers.

W-4C-3. MULTI-SCALE ENERGY TRANSFER AND FLOW PATTERN

Z. Y. WANG, H. T. JIA, C. X. XU and G. X. CUI, *School of Aerospace, Tsinghua University, Beijing, China,* The investigation into the relationship between the multi-scale energy transfer and near-wall coherent structures is helpful for the understanding of the physics and the development of the models of wall turbulence. To yield a more general scenario, we studied the relationship between the multi-scale energy transfer and flow patterns in turbulent channel flow with the aid of critical-point theory. The present work combines the conditional averages of SGS dissipation term and the classification of flow patterns by second and third invariants of velocity gradient tensor, Q and R, and the discriminant of its characteristic equation, D. By using the DNS data of turbulent channel flow at $\mathrm{Re}_\tau = 395$, the relationship between multi-scale energy transfer and flow patterns are elaborated in more detail. It is found that for forward scatter of turbulent kinetic energy, the unstable node-saddle-saddle pattern is dominant, while for backward scatter, all the flow patterns have the similar probability. The ratio for all the four flow patterns in both forward and backward scatter events changed strongly in viscous sub-layer and in buffer region, and it becomes almost constant further away from the wall. The flow fields around strong forward and backward scatter events are obtained in logarithmic region, and it is shown that the multi-scale energy transfer is not only related with vortex structures but also with node-saddle structures, and we can see from the distribution on the (R,Q)-plane of the velocity field on (x,z)-plane at $y^+ = 46.72$, the distribution for forward scatter events converges to small but negative D, which represents unstable node-saddle structure, for backward scatter events, most points distribute on the region for the stable foci structure, which is in accordance with Natrajan (Phys. Fluids, 2006).

W-4C-4. EJECTIONS AND BURSTS IN A LOW DRAG REDUCTION TURBULENT CHANNEL FLOW OF DILUTE POLYMER SOLUTIONS

C.-F. LI, G.-F. WU, X.-D. FENG, Z.-G. ZHAO, *Jiangsu University, Zhenjiang, China,* R. SURESHKUMAR, *Washington University, St. Louis, USA,* B. KHOMAMI, *University of Tennessee, Knoxville, USA,* Ejections and bursts in turbulent channel flow of dilute polymer solutions have been studied via direct numerical simulations and quadrant analysis. In this paper the drag reduction levels (%DR) in the low drag reduction regime (LDR, 0 <%DR< 30–40%) have been mainly investigated where bursting events can be clearly distinguished. It has been seen that both in the near wall region and in the core the average time interval between bursts increases as drag reduction is enhanced. With the fixed maximum chain extensibility the increase in the time interval between bursts is shown to be directly related to the average polymer chain extension in the flow. Specifically, the enhanced chain extension gives rise to enhanced extensional viscosity of the polymeric solution, thereby stabilizing streamwise vortices and giving rise to elongated axial vortices. This in turn increases the average time between bursts in the near wall region resulting in a reduction of overall turbulent production (i.e., drag reduction) via suppression of the bursts. To shed light on the exact mechanism by which polymer induce drag reduction occurs the various time scales of the flow have been examined. A simple framework is proposed adequately to describe the influence of polymer additives on all drag reduction regimes (from onset to the Virk asymptote - maximum drag reduction, MDR) as well as the universality of the MDR in flow systems.

15:00 ~ 16:20 (Room104)

Aerodynamics (VI)