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)