Tuesday, August 19

Plenary Lectures & Invited Lectures

PL-3. MODELLING OF HIGH-SPEED TURBULENT AND TURBULENT-TRANSITION FLOWS WITH RANS APPROACH
Song FU, *Department of Engineering Mechanics, Tsinghua University, China,* High-speed turbulence flows in engineering applications are often characterized with two fundamental flow features: the effect of compressibility and the effect of large velocity gradients. It is well known that compressibility effect plays an important role in determining the turbulence physics when Mach number Ma is high and the conventional Mokovian hypothesis leads to significant errors in modelling. It is also known that the compressibility strongly influences the behaviour of supersonic/ hypersonic flow transition compared to low-speed case. The large velocity gradients in the flow, either in form of shear or strain, generally lead to rapid flow distortion to turbulence that requires delicate modelling practice. However, current compressible turbulence/transition models are still not able to provide satisfactory results to meet the urgent engineering demands. Existing turbulence models also fail to capture the physics of rapidly distorted turbulent flows which, in the limit, satisfy Rapid Distortion Theory (RDT). This article provides an overview of some of the research efforts in turbulence modelling made at Tsinghua University over the last few years in an attempt to improve predictive capabilities for flows associated with significant compressibility and distortion effects. In particular, three areas will be primarily focused on: a) Modelling of hypersonic boundary-layer transition flows; b) Modelling of the compressibility effect with second-moment closure; c) Modelling the rapid pressure-strain correlation satisfying RDT. In this work, a new k-ω-γ transition/turbulence model considering the modes of instability is proposed and validated. The model is based on local variables and is able to trigger the onset of transition inherently with the function in the source term of γ equation. The present model has been successfully applied to simulate the natural, as well as the bypass transitions. A new second-moment closure model is also proposed here and applied to the simulation of compressible mixing layers. The calculated growth rates agree reasonably well with the 'Langley curve'. A new approach to improving the prediction of the anisotropy evolution in rapid limit, where the M -tensor of rapid pressure-strain correlation is expandable in the Reynolds-stress anisotropy tensor and mean rotation rate tensor. This extension of FLT model allows two different traces in Anisotropy Invariant Map for plane strain and shear flow, and a damped oscillatory solution for the anisotropy components in the situation of pure rotation, which are not even qualitative captured by classical rapid pressure-strain models. Present work presents a possible way to extend classical model to rapidly deformed flow field.

PL-4. DYNAMICS OF SHEARED GRANULAR FLUID
Meheboob ALAM, *Engineering Mechanics Unit, Jawaharlal Nehru Center for Advanced Scientific Research, India,* The dynamics of sheared granular fluid is briefly reviewed, focusing on instabilities, patterns and bifurcations in plane shear flow. It is shown that a universal criterion holds for the onset of shear-banding instability (for perturbations having no variations along the stream-wise direction) that leads to shear-band formation along the gradient direction. The same shear-banding criterion appears to hold in other complex fluids as well as in the singular limit of atomistic fluids (i.e., elastic hard-spheres). A weakly non-linear analysis of the shear-banding instability unveils that the lower branch of the neutral stability curve, that corresponds to dilute flows, is sub-critically unstable. In the presence of gravity, the origin of such shear-banding transition is shown to be tied to the spontaneous shear-banding instabilities of the gravity-free uniform shear flow, resulting in universal unfolding of pitchfork bifurcations in gravity-modulated plane shear flow.

PL-5. SOME ASPECTS OF VORTEX ASYMMETRY AND ITS CONTROL ON SLENDERBODIES AT HIGH ANGLES OF ATTACK
P. R. VISWANATH, *Department of Aerospace Engineering Indian Institute of Science, India,* The problem of vortex asymmetry and the associated side forces (and yawing moments) on pointed forebodies at high angles of attack and zero side slip has received considerable attention in literature. Beyond a certain angle of attack (depending primarily on the nose apex angle), the symmetric vortex flows become asymmetric resulting in side force generation; the side force generally increases with α and reaches a maximum typically in the α range of 40-50 deg. This regime of three-dimensional separated flow over a pointed slenderbody is nominally steady and the forces / moments generated are generally repeatable for a given model and given roll position in a wind tunnel. Our broad knowledge of this complex flow involving vortex asymmetry has been the result of extensive experimental research on axisymmetric bodies during the last three decades, which have revealed the important parameters affecting the onset of vortex asymmetry and the magnitude of side forces. The side forces generated are strongly Reynolds number dependent and the effects gradually decrease with increase in flight Mach number – the problem is essentially predominant at low to subsonic speeds in which regime the high alpha maneuvers of combat aircraft and missiles normally occur. Excellent reviews on the subject have been published over the years and the problem is still intractable to modeling and predictions, even in an engineering sense. In this lecture, we present a brief review highlighting some of the developments that have taken place in the broad understanding of this complex phenomenon and control of side forces, which are very important from a technological viewpoint. Several papers in literature have focused attention on the cause of asymmetry and have attempted to provide explanations based largely on inviscid arguments. It appears that geometric imperfections / micro-asymmetries in the nose region have a significant influence in triggering vortex asymmetry. Several side force control methods have been explored in literature which may have future applications. Selected results from two recent studies (carried out at the National Aerospace Laboratories, Bangalore) will be presented showing the effectiveness of nose bluntness and axial nose blowing for side force control.

PL-6. MULTISCALE ANALYSIS OF BLOOD FLOW: MODELING AND SIMULATION OF MULTIPLE RED BLOOD CELL FLOW
S. WADA and M. NAKAMURA, *Osaka University, Japan,* Blood is an inhomogeneous fluid consisting of blood cells suspended in a liquid component called plasma. The most abundant cells are red blood cells (RBCs), which occupy almost half of the whole blood volume. Therefore, the fluid properties of blood are influenced by the concentration (hematocrit), elasticity, and aggregation of RBCs and the viscosity of plasma. In this study, we developed a three-dimensional model of an elastic RBC based on the minimum energy principle, and simulated the dynamical behavior of multiple RBCs in flowing blood. The RBC model was constructed by surrounding the internal liquid with a spring network that represents elastic resistances to stretching, bending, and area expansion of the RBC membrane. The mechanical interaction among multiple RBCs was expressed as a potential function discretely assigned at the membrane boundary. Using the momentum conservation law and Newton's friction law, the fluid forces acting on the membrane were estimated from the difference in the velocity between the RBC movement and the theoretical fluid flow. First, the behavior of a single RBC in a shear flow was investigated. The RBC was rotated as a rigid body at a shear rate of less than 20 s^{-1}, while it was orientated at a constant angle by rotating its membrane at a higher shear rate. These results were consistent with experimental observations. Second, the behavior of multiple RBCs in a small artery with a Poiseuille flow was simulated using the Earth Simulator—a highly parallel vector supercomputer. We allocated 16,256 RBCs in a straight artery with a diameter of 106 μm and a length of 1024 μm. The simulation showed the formation of a plasma skimming layer near the vessel wall. The RBCs exhibited tumbling and tank-treading motion about the central axis and near the wall, respectively, depending on the shear rate of the flow. Finally, we proposed a novel simulation method for performing multiscale analysis of blood flow. The macroscopic flow was modeled by a continuum described by the continuity and Navier–Stokes equations. The axial velocity profile of the macroscopic flow was used to calculate RBC behavior, as mentioned above. To consider the influence of the microscopic behavior of RBCs on the macroscopic flow, hematocrit-dependent viscosities estimated from the RBC distribution were used in the macroscopic flow calculation. Cylindrical tube flow at a Reynold's number of 0.1 was simulated by interactively repeating these macro and microscale analyses. Results showed that RBCs migrated axially, and increased hematocrit around the central axis of the flow channel while decreasing it near the wall. Consequently, velocity tended to be lower around the center while that near the wall increased. At the converged state, the velocity profile was blunt, as observed *in vivo*. These results indicated the potential of the present computational method in the analysis of blood rheology.

IL-5. NUMERICAL ANALYSIS OF AERODYNAMIC NOISE FROM FEEDBACK PHENOMENA USING COMPUTATIONAL AEROACOUSTICS (CAA)
D. J. LEE, *KAIST, Korea*, I. C. LEE, *KAIST, Korea*, D. N. HEO, *KISTEP,*

Korea, Y. N. KIM, *Flow-Noise Company, Korea,* Computational Aeroacoustics (CAA) deals about capturing radiated acoustic quantities generated from flow fluctuations numerically. In general, the amplitude of acoustics is less than 4^{th} order of the flow. Therefore, a higher order scheme, such as compact scheme, is employed to capture the acoustic and flow at the same time. To minimize the numerical phase error in the radiated acoustics, the coefficients of the high order scheme are optimized to have minimum dispersion error. The high order optimized compact schemes are applied in the acoustic propagation of the cavity tone from the subsonic flow and screech tone from the supersonic flow. Cavity tone and Screech tone are generated due to the feedback between flow and acoustic wave. In this paper, the feedback phenomena are calculated numerically to obtain detail information of flow and acoustic wave to explain the mechanism including the phase shift and mode change. The detail calculation is used for the time required for the feedback and phase problem. It is found that no phase shift is required if we examine the time required carefully. The phase shift of cavity tone is depending on the position of the acoustic source and the mode of the tone. Rossiter's formula for the cavity tone used for quick explanation of mode from experimental data needs to be reexamined. Screech tone is also calculated with the high order high resolution scheme. The tone is due to the feedback between the flow and acoustic and the numerical results are compared with experimental data for the Mach number of mode change, which shows reasonable agreements. Also the transient characteristics of axisymmetric screech tones are investigated and three dimensional screech tone is also simulated briefly. Even limited physical and numerical conditions in calculation because of high order high resolution scheme, the phase problem can be clearly explained for the cavity in the range of laminar cases and the mode change Mach number is reasonably predicted with the inviscid assumption for the axisymmetric supersonic jet. Additionally, other effective methods for numerical analysis of incompressible flow noise are addressed and discussed.

IL-6. PROGRESS IN THE DEVELOPMENT AND APPLICATION OF LATTICE BOLTZMANN METHOD

C. SHU, Y. T. CHEW, X. D. NIU, Y. PENG, H. W. ZHENG and N. Y. LIU, *Department of Mechanical Engineering, National University of Singapore, Singapore,* As an alternative computational fluid dynamics approach, lattice Boltzmann method (LBM) receives more and more attention in recent years. LBM is a particle-based approach, which does not involve the solution of partial differential equations and their resultant algebraic equations. Thus, its implementation and coding are very simple. Currently, LBM has been widely applied to simulate various fluid flow problems. This paper will report the progress in the development and application of LBM by the group of National University of Singapore. In the development of lattice Boltzmann model, we developed a general platform for the users to design their own lattice velocity model and associated equilibrium distribution functions. In the application of LBM, we developed the Taylor series expansion- and least-square-based lattice Boltzmann method (TLLBM), simplified thermal lattice Boltzmann model, lattice kinetic scheme and the fractional step lattice Boltzmann model. These models can effectively simulate isothermal and thermal flows with complex geometry and at high Reynolds numbers. In the application of LBM for simulation of micro flows, we proposed a new relationship between relaxation parameter τ and Knudsen number, and the diffuse-scattering boundary condition (DSBC) from the kinetic theory. In the application of LBM for simulation of multiphase flows, we presented a new interface capturing lattice Boltzmann model, which can recover the Cahn-Hilliard equation up to the second order of accuracy. The proposed model can well simulate multiphase flows with large density ratio. Recently, we developed a new lattice Boltzmann model for simulation of compressible flows with strong shock waves. The equilibrium distribution functions and associated lattice velocity model are developed from satisfaction of conservation laws in physics. The paper will also address our latest development of lattice Boltzmann-immersed boundary velocity correction method (LB-IBVCM), which can accurately satisfy the non-slip condition on the wall.

IL-7. HYDRAULIC MODELING OF SOIL EROSION

Q. Q. LIU, *IMECH CAS, China,* The prediction and estimate of soil erosion is fundamental important for understanding the effect of the spatial heterogeneity of underlying surfaces and preventing ecological environment deterioration. Since soil and rainfall characteristics substantially vary in different regions, the empirical models do not reflect the overall effect of various factors. Accordingly, there seems to be a shift in emphasis from the empirical approach to the process-based dynamic approach to soil erosion. The water erosion is mainly caused by natural rainfall, and is such a process that sheet flow generated during rainfall scours the soil surface. The erosion process can be divided into three basic dynamics processes, including the process of runoff generation caused by rainfall, the process of sediment yield on hillslope by overland flow, and the process of runoff concentration and sediment transport on watersheds. A process-based soil erosion model was developed according to the characteristics of soil erosion on the Loess Plateau. The proposed model includes three component models: the rainfall-runoff sub-model on hillslopes, the soil erosion sub-model on hillslopes and the runoff concentration and sediment transport sub-model on watersheds. The kinematic wave approximation combining the infiltration excess runoff was applied to describe the runoff yield process. Interrill erosion and rill erosion are two basic types of soil erosion on rural hillslopes. Therefore, the soil erosion sub-model includes these two parts: interrill erosion and rill erosion. A two-dimensional hydrodynamics model was employed to describe the runoff concentration and sediment transport. The erosion model on hillslopes was verified by laboratory experiments, and overall, good agreements were found between simulation results and experimental observations. Rainfall and slope characteristics affecting runoff generation and soil erosion on hillslopes were analyzed by using the proposed model. The primary hydraulic characteristics of the runoff generation, such as unit discharge, runoff depth, flow velocity, shear stress and ratio of runoff generation are obtained and analyzed. Especially, the slope length and gradient play important roles in the processes of soil erosion on hillslopes. The modeling results show that the slope length and gradient, time distribution rainfall, and distribution of rills have varying influence on soil erosion. Erosion rate increases nonlinearly with increase in the slope length; a long slope length leads to more serious erosion. The effect of the slope gradient on soil erosion can be both positive and negative. Thus, there exists a critical slope gradient for soil erosion, which is about 25^o for the accumulated erosion. Applying the proposed model to a typical small catchment in the loess plateau area of China, the runoff and sediment yield process was estimated, which exhibited a good agreement between predicted results and observation. It also demonstrated that the proposed model is capable of adequately simulating the process of runoff yield and soil erosion on small watersheds.

IL-8. SIMULATION, PREDICTION AND EXPERIMENT ON WINDBLOWN SAND MOVEMENT AND AEOLIAN GEOMORPHOLOGY

X. J. ZHENG, *Key Laboratory of Mechanics on Western Disaster and Environment, Lanzhou University, China,* In the evolution processes of wind blown sand movement and aeolian geomorphology, it always contains some complex behaviors, for example, the nonlinear character of turbulence and attractors, the stochastic character of wind gust, liftoff and movement of sand, the interaction among wind field, sand movement, electric field in wind blown sand flux and thermal diffusion, multi-scale character from sand ripple to dune, which deserve to be paid attention by mechanical researchers. In this paper, we introduce the recent works of our research group in Lanzhou University, China on the measurement, modeling and simulation of wind blown sand movement and aeolian geomorphology in detail.

10:30 ~ 11:50 (Room 101)

Reacting Flows (I)

Session Chair : Prof. I. Lee, Pusan Univ/Korea

T-1A-1. NUMERICAL SIMULATION OF A 200 MW INDUSTRIAL BOILER

M. D. EMAMI, *Isfahan University of Technology, Isfahan, Iran*, S. POURARIAN, *Nargan Engineering Co., Tehran, Iran*, H. AFSHIN, *Sharif University of Technology, Tehran, Iran*, S. ZIAEI-RAD, *Isfahan University of Technology, Isfahan, Iran,* Numerical simulation of a gaseous-fuelled boiler of a power plant has been performed, using Favre-averaged equations of mass, momentum, energy, turbulent kinetic energy and its dissipation, the transport equations of the mixture fraction and its variance, and the radiation transfer equation. The mixture fraction concept is used to model combustion, and the turbulence-combustion interaction is taken into account by the use of a presumed probability density function. The computer code is a finite volume based code, with collocated grids and SIMPLEC algorithm. Higher-order convection schemes and second-order diffusion schemes are used for discretization of the governing partial differential equations. The purpose was to find out the reason for overheating the super-heater tubes of the boiler, and proposing a remedy for the problem. Results of the base case show hot regions in the aerodynamic nose of the boiler, which are undesirable because of proximity of super-heater tubes to this area. The Nitrogen mass fraction contours, which are measures of the air distribution, also reveal non-uniformity in air