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 4th 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 25º 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 distribution in this region. Two strategies may be followed to modify the combustion phenomenon in this boiler. The first one is to change the swirl angle of the inlet air to prevent the elongation of flame towards the aerodynamic nose. The second approach, which is adopted in this study, is to change the ratio of primary to secondary air flow rate. Numerical experiments show that a ratio of 40% to 60% primary to secondary air instead of the present 30% to 70% ratio would result in better uniformity in the temperature profile and air distribution at the aerodynamic nose and limits the maximum temperature region.

T-1A-2. HIGHER-ORDER SPEED GRADIENT VISCOUS CONTINUUM MODEL

H. X. GE, *Facultyt of Science, Ningbo University, Ningbo, China,* In the light of the microscopic two velocity difference model, a new macroscopic model called speed viscous continuum model is developed to describe traffic more reasonably. The relative velocities are added to the motion equation, which leads to viscous effects in continuum model traffic flow dynamics. The qualities of the new model are investigated in detail. The viscous continuum model overcomes the backward travel problem, which exists in many higher-order continuum models.

T-1A-3. NUMERICAL STUDY OF THE EFFECT OF INLET PARAMETERS ON THE FLASH-BACK LIMIT OF A POROUS BURNER

P. RIAHI, M. H. AKBARI, *Shiraz University, Iran,* Combustion in porous media has many advantages in comparison with free flame combustion. This technique enhances the efficiency of a combustion system and offers higher flame speeds and power densities, stable combustion for a wide range of equivalence ratios, higher dynamic power ranges, high compactness and less emission than free flame combustion. Considerable efforts have been made to demonstrate the practical benefits of porous medium burners. However, the study of flash-back limits has received little attention in the literature. In this work, submerged laminar premixed flame propagation of methane in an inert homogeneous Cordierite (with LS2) matrix is numerically investigated. For this purpose, an unsteady one-dimensional physical model of the porous burner, using a one-step global chemical kinetics is considered. Continuity, species conservation and thermally non-equilibrium (separate) solid and gas energy are the equations which govern this problem, and are derived by the spatial averaging method. Gas mixture is treated as an ideal gas and all its thermophysical properties are taken as functions of the temperature. Gas phase radiation is neglected, while radiative heat flux in the solid matrix is modeled using a diffusion approximation. The computational domain is extended beyond either side of the porous region to accurately model reactions close to the edges of the porous region. For this purpose three distinct regions A, B and C are considered. The solid matrix is confined in region B, and regions A and C contain only fluid phase. The model dimensions, as well as the initial and boundary conditions, are set such that the model makes physical sense. The governing equations are discretized using a fully implicit finite volume method. The resulting algebraic equations are solved by the Tri-Diagonal Matrix Algorithm. A relative convergence criteria for numerical computation of all variables is set to 1×10^{-5}. After a baseline simulation, the influence of three inlet parameters, including the inlet temperature, equivalence ratio, and inlet firing rate on the burner thermal performance and the flame flash-back limit in region B are investigated. The flame temperature increases, and its location moves upstream with an increase in the inlet temperature of the reactants or the mixture equivalence ratio. Decreasing the inlet firing rate will decrease the flame temperature to some extent. The simulation results show that the combustion products temperature will rise by increasing the inlet temperature or the inlet firing rate, but an increase in the equivalence ratio will result in a slightly cooler product formation. The solid phase temperature becomes more uniform with an increase in the mixture inlet temperature or the equivalence ratio or a reduction in the inlet firing rate. The flame displacement towards the solid upstream may result in flash-back which is a very undesirable phenomenon in the operation of a porous burner. Based on such simulations, the influence of the studied inlet parameters on the flame flash-back limits are investigated.

T-1A-4. NUMERICAL COMBUSTION MODELING OF A GAS-BURNER AND STUDYING ITS EFFECTING PARAMETERS

A. KIANIFAR, N. GHAFOORIANFAR, H. MOIN, I. R. TOROGHI, M. JAVADI *Ferdowsi University of Mashhad, Iran,* In this paper numerical simulation of combustion over a sample of a prevalent gas-burner, and effects of parameters such as environment geometry, main parameters of chimney, effects of free and forced convection on the environment, in radiation modeling situation and without radiation has been studied. Simulation of combustion process with the purpose of studying the amount of pollutants produced by combustion needs perfect identification of this phenomenon from chemical point of view, thermodynamic point of view, and fluid mechanics point of view. The best case for inflammation complex of natural gas is to observe fuel-air ratio 1 to 10. But if there is not enough air available, or the flame is not complete and uniform, there is not enough time for carbon monoxide to oxidize and convert to carbon dioxide and will be released. The flow regime in the combustion chamber under studied is turbulent with change in density of the chemical species, which is aroused from the combustion. The governing equations on this phenomenon are conservation of mass, momentum, transmission of species and energy in the cylinder coordinate system with the assumption of steady with respect to time. For modeling the terms aroused from turbulent assumption k-ε method and for modeling the combustion flow and calculating transmission of species Eddy-Dissipation method has been used. For calculating the turbulence effects on the properties of flow and calculating the effective heat conduction factor and effective viscosity two assistance equations (k&ε) has been utilized.

In this study combustion of methane-air assumed with two stage combustion mechanism is used. On the basis of this mechanism, the products of methane oxidization are carbon monoxide and water vapor. In the next stage carbon dioxide formed from carbon monoxide oxidization. In this modeling, entrance surface of fuel and air is considered to be a part of a cylindrical tube with a determined cross section. In this paper gas-burner is studied in two different ways: 1- Modeling of gas-burner and its surrounding in order to study atmosphere effects on efficiency of the gas-burner. 2- Modeling of gas-burner and imposing atmosphere effects on gas-burner walls with imposing convection heat transfer boundary conditions.

The results show that the amount of heat transfer will increase with increase in gas-burner height. Also increasing in gas-burner width leads will increase the amount of heat transfer. Modeling of this heating device in two geometric environments, which are room with four walls, and room with three walls and one open surface, which are the most usual uses of gas-burner, discloses advantages and disadvantages, temperature distribution and heat transfer trend in each of these environments. Nondimensional temperature distribution is a massive help to compare the gas-burner efficiency in these two environments.

10:30 ~ 11:50 (Room 102)

Experimental Techniques (I)

Session Chair : Prof. M. Princevac, California Univ/USA

T-1B-1. PIV MEASUREMENTS ON AN AIR BLAST ATOMISER

P. SURIYANARAYANAN, *National Aerospace Laboratories*, *India* and L. VENKATAKRISHNAN, *National Aerospace Laboratories*, *India,* The flow field of a typical airblast injector was studied using PIV. The air-water flow field was documented with 2D PIV and the air-alone flow at four streamwise locations with stereo PIV. The results show that for a fixed air-water mass ratio, volume flow rate has a negligible effect on the dispersion angle, but a considerable effect on the mean velocity field. The stereo PIV measurements indicate the presence of a region of reversed flow at the injector centerline which initially increases and persists downstream though considerably reduced in size and magnitude. This is because the large swirl angle of the air slots creates a highly swirling flow which sets up a recirculation zone due resulting in the formation of a toroidal vortex near the exit. The findings have significant implications for combustor design and can be used to optimize airblast injectors for efficient fuel mixing.

T-1B-2. MICRO HOLOGRAPHIC PTV MEASUREMENTS OF DEAN FLOWS IN A CURVED MICRO-TUBE

S. KIM, *POSTECH, Korea,* S. J. LEE, *POSTECH, Korea,* In the present study, a micro holographic PTV (HPTV) system was used to experimentally investigate the structure of 3D flow within a curved microtube with varying Dean Number. The employed HPTV system incorporated a high-speed digital camera to measure the temporal evolution of the 3D velocity fields of micro-scale fluid flows. In this study, to analyze the 3D flow characteristics in the curved section of tube at a high Dean number, the trajectories of fluid particles were obtained experimentally using the whole 3D velocity field data obtained by the micro HPTV technique. These results would be helpful in the design of various passages within micro-scale devices or micro-chips and in understanding the mixing phenomena that occur in curved conduits along the trajectories of fluid particles. The HPTV system consists of a high-speed digital camera, a laser, an AOM chopper, and a mirror. A He-Ne laser (λ=633nm) was used as a light source, and a