T-1G-3. ON THE ROBUSTNESS AND ACCURACY OF LEAST SQUARES KINETIC UPWIND METHOD (LSKUM)

Konark ARORA, *Research Associate, EMU, JNCASR, Jakkur, Bangalore, India,* N. K. S. RAJAN, *CGPL, Dept. of Aerospace Engg., IISc, Bangalore, India,* S. M. DESHPANDE, *EMU, JNCASR, Jakkur, Bangalore, India,* The gridfree methods do not require the grid for flow simulation, which is an essential requirement for all other conventional CFD methods. But they do require a point distribution or a cloud of points. However some problems are encountered while using this method on point distributions around complex configurations. Close analysis of this problem has revealed that bad connectivity of the nodes is the cause leading to code divergence. To overcome this problem of bad connectivity related code divergence, we have followed a novel approach of using weights to alter the nature of the least squares (LS) matrix A. It is observed that along the eigendirections, the corresponding LS formulae reduce to the 1-D type formulae. But a problem arises in combining the eigendirections along with upwinding, which is essential to provide stability of the numerical scheme. This difficulty has been overcome by the use of weights leading to WLSKUM-ED (Weighted Least Squares Kinetic Upwind Method using Eigendirections) wherein the weights are suitably chosen so that a chosen direction becomes an eigendirection of matrix A(w) along which multidimensional LS formulae reduce to the corresponding 1-D type formulae. Higher order accuracy in the LS gridfree methods is obtained by defect correction method. The defect correction based second order accurate LS formula has an implicit dependence on the value of the derivatives at the neighbouring nodes. Hence a few iterations need to be performed to get the second order accurate value of the derivatives. These iterations are called the inner iterations. It has been shown that the formula for the derivative in LS gridfree methods reduces to the standard Finite Difference formula in case of a uniform point distribution. Thus it is quite easy to obtain the formal order of accuracy in case of uniform point distribution. However, in case of an arbitrary point distribution, it is essential to perform some inner iteration to get the desired formal order of accuracy. Combination of WLSKUM-ED with inner iterations has been shown to lead to kinetic meshfree method which is both robust and accurate**.**

T-1G-4. NUMERICAL STUDY OF FLOW-INDUCED OSCILLATIONS OF A CIRCULAR CYLINDER USING OVERSET GRID METHOD

M. SAGHAFIAN, F. BARATCHI, *Isfahan University of Technology, Isfahan,, Iran,* In this paper, a numerical study of a uniform flow over an elastic and responding circular cylinder is presented. The Overset grid approach is applied to prevent remeshing procedure. In this method a major rectangular grid and a minor body fitted O-type co-located grid over circular cylinder (located in the rectangular grid) are used. The flow is solved separately on each mesh or grid and the solutions variables are transmitted from one mesh to the other mesh by interpolating in certain points that are called fringe points. The directional search algorithm is used to localize the fringe points of one grid in the other grid sequentially. Finite volume approach is applied to discritise the differential equations. SIMPLEC algorithm is employed to solve the discritised equations. The QUICK scheme is applied in calculating the convective terms. The Crank-Nicholson time marching method is employed and Rhie-Chow method is used to prevent non-physical oscillations. The Reynolds number is set to 200 and a spring-damper-mass system is used to model the cylinder motions. The cylinder responses and the structure of vortex shedding have been analysed. The effects of dimensionless mass and damping coefficient has been investigated. It is observed that, when structural frequency is near to frequency of vortex shedding of stationary cylinder at the same Reynolds number, lock in will occur. In this case, the drag coefficient, the displacements and the amplitude of oscillations in longitudinal and transverse directions attain their maximum values. In addition, the amplitude of oscillations could be as large as 65 percent of the cylinder diameter for small mass and damping coefficient which a dimensionless structural frequency is near 0.2. In overall, the mean value of the lift coefficient is nearly zero; however, its amplitude size is considerable. In comparison, the mean value of the drag coefficient is significant particularly when lock in occurs. However, its amplitude is small. Thus, the displacement in longitudinal (x) direction is larger than in the transverse (y) direction. But, the amplitude of oscillations in (x) direction is smaller than the amplitude of oscillations in (y) direction.

14:30 ~ 15:50 (Room 101)

Reacting Flows (II)

Session Chair : Prof. P. R. Viswanath, IIS/India

T-2A-1. PIV OF MODE SHIFTS DURING COMBUSTION-ACOUSTIC LOCK-ON IN A NON-PREMIXED BACKWARD-FACING STEP COMBUSTOR

O. J. SHREENIVASAN and S. R. CHAKRAVARTHY, *IIT Madras, India,* This paper reports acoustic mode shifts during combustion-acoustic lock-on in a rectangular non-premixed backward-facing step combustor, in which methane enters the combustor at the corner of the step, and mixes and burns with the air flowing past the step in the unsteady recirculation zone downstream. The investigation is carried out for the Reynolds number range of 6000-60000. The interaction of the flow field and the flame is investigated using phase-locked PIV and high-speed CH* chemiluminescence imaging, along with simultaneous unsteady pressure measurement. As the flow Reynolds number is increased, the dominant frequency in the unsteady pressure spectra converges to a constant value around the no-heat-release natural acoustic frequency of the combustor duct, and then abruptly jumps to a regime of linear increase in frequency with increase in Reynolds number, signifying the onset of a first lock-on. A subsequent jump to another regime of linear increase with a lower slope signifies the second lock-on. The transition from the constant-frequency regime to the linearly increasing one corresponds to a rise in amplitude from low broadband levels to high discrete tones. The subsequent jump from one linear variation to another is also accompanied by a further rise in the amplitude. The phase-locked ensemble-averaged PIV shows flow structures of different length-scales at different phases in the pre-lock-on regime, but a dominant large-scale roll-up during the lock-on regime. The leading edge of the flame, observed in the phase-averaged CH* chemiluminescene images from high-speed imaging, moves in and out of the vortex core over the oscillatory cycle. The chemiluminescent intensity also fluctuates significantly over the oscillatory phase. On the contrary, the flame is located downstream of the recirculation zone and its chemiluminescent intensity fluctuates little in the pre-lock-on regime. These observations explain the mechanism of mode shifts that leads to combustion instability in a backward-facing step combustor.

T-2A-2. CFD SIMULATION OF A GAS TURBINE FOR EXPERIMENTAL PURPOSE

C. F. CHENG and C. L. YOONG, *Department of Mechanical Engineering, Curtin University of Technology Sarawak Campus, Malaysia,* A CFD study has been performed to simulate the gasses combustion within a combustion chamber. A 3D can-type combustion chamber was considered with the simulation performed using FLUENT. Propane (C_3H_8) was used as the combustion fuel agent. Standard k-e turbulent species transport (non-premixed) combustion model with Eddy Dissipation model is used to simulate the combustion process. Four species of simulation were considered, i.e. oxygen (O_2), carbon dioxide (CO_2), C_3H_8 and water (H_2O). The simulation results enable the visualization of flow behaviour of combustion agents within the combustion chambers, i.e. velocity, pressure and temperature. Species concentration and distribution was observed in the combustion process. It is observed that higher temperature profiles occurred at the exit region of combustion chamber. Also found, that the geometrical shape of combustion chamber plays an important role in maximizing the flow parameters by reducing undesirable flow resistance. The simulation result provides a useful guideline in the later stage to design and construct of a simple gas turbine prototype for experimental purpose.

T-2A-3. USE OF SWIRL TO CONTROL OSCILLATIONS IN LEAN COMBUSTION

P. GEIPEL, *Imperial College London, U.K.*, R. P. LINDSTEDT, *Imperial College London, U.K.*, S. SIVASEGARAM, *University of Peradeniya, Sri Lanka*. Extinction in lean premixed flames is preceded by a series of cycles of extinction and relight during which the flame gradually weakens until its final blow-out. Such cycles of extinction and relight can also occur over a range of equivalence ratios close to the lean flammability limit to give rise to large amplitudes of oscillation, thereby narrowing the stable range of operation of the combustor. These low frequency oscillations at around 10 Hz, manifesting themselves as a modulation in amplitude of the wall static pressure signal, are not amenable to active control owing to the broadband nature of the frequency; and the problem is further aggravated by stratification of the fuel-air mixture in ways similar to that in gas turbine combustors. The possibility of ameliorating the oscillations by improving flame stabilization using the addition of swirl and thereby reducing the tendency for local extinction was examined in a round sudden expansion combustor burning a stratified mixture of methane and air. Stratification was achieved using a relatively rich core flow and a leaner annular flow around it in the duct section upstream of the expansion; swirl was added to the upstream core flow or alternatively to the annular flow, using a fixed-

vane swirler. The addition of swirl, while improving flame stability, also increased local strain rates in the region of flame stabilisation so that the lean flammability limit increased with swirl number. Moderate rates of swirl added to the core or to the annular flow were, however, beneficial and eliminated the low frequency oscillation without a significant increase in the lean flammability limit. While swirl reduced the low frequency modulation in amplitude of the pressure signal, at times, it gave rise to a frequency of the order of 20 Hz of small amplitude. This frequency increased with bulk flow rate and swirl number, indicating that it was associated with the recirculation zone adjoining the sudden expansion, whose length decreased with swirl. Larger rates of swirl were not beneficial and led to a higher lean flammability limit as well as to an increase in the amplitude of oscillations associated with the natural acoustic frequencies of the combustor duct.

T-2A-4. MODELING OF URBAN CANOPY FLOWS IN A WATER CHANNEL

Marko PRINCEVAC, Xiangyi LI, and Hansheng PAN, *Department of Mechanical Engineering, University of California, U.S.A,* Detailed measurements of the flow within the modeled urban roughness sublayer were conducted in the water channel at the University of California, Riverside, and Laboratory for Environmental Flow Modeling. Building configurations simulating simple urban patterns were accomplished using highly polished acrylic models to minimize effects of refraction and attenuation of the laser sheet utilized by the TSI Particle Image Velocimetry (PIV) and the Planar Laser Induced Fluorescence (PLIF) systems. First, using a simple two building configuration the effects of channeling were studied and distribution of the turbulent kinetic energy was measured. Flow approach angle was 1, 3, 5 and 7 degrees and the investigated ratios of building heights to the street width were 0.5, 2 and 4 corresponding to the skimming flow, wake interference, and isolated wake regimes, respectively. The occurrence of flow channeling vs. flow recirculation in the street canyon was observed and the criteria for channeling occurrence was established. Second, flow within a simple 3 by 3 and 5 by 5 cubical building arrays were studied. This is the first time that such detailed measurements of the flow between the obstacles were performed. It was found that lateral array size has significant influence on initial dispersion within the array. Smaller array size leads to sideways flow channeling causing significant plume spread. This sideways channeling becomes less pronounced as array size increases. The sideways channeling becomes more intense when the mid-array building is higher.

14:30 ~ 15:50 (Room 102)

Experimental Techniques (II)

Session Chair : Prof. O. Mochizuki, Toyo Univ/Japan

T-2B-1. EXPERIMENTS WITH REAL SOURCE - SINK PAIRS

Sudhakar SUBUDHI, Jaywant H. ARAKERI, *Indian Institute of Science, Bangalore, India,* K. R.SREENIVAS, *Jawahar Nehru Center for Advanced Scientific Research, Bangalore, India,* This paper deals with the flow associated with a source and a sink. Such source-sink interactions occur in many situations including cooling of computer data centers. The source consists of fluid issuing out of a pipe and the sink is a pipe, through which fluid enters, that is kept some distance from the source pipe. The source and sink flow rates may not be same. Of concern is the percentage of source fluid that enters the sink. Experiments have been carried in a tank of size of 1200mm×430mm×415mm with its top side open to ambient. The working fluid is water. The source pipe ID is of 6mm and the sink pipe ID is of 10mm or 20mm. The horizontal and vertical separations between the source and sink pipes are adjustable. There are three types of geometries considered: (a) when source and sink have horizontal separation, (b) when source and sink have horizontal and vertical separations and (c) when source pipe and sink pipe are at right angles with each other. The Reynolds number (Re) was about 3200 based on the exit diameter of the source. This means that the jet was turbulent. Experiments were done with the sink flow rate equal to , lower or higher than the source flow rate . Potassium permanganate dye is used for flow visualization and from this visualization, the approximate flow rate of the sink for which all the source fluid was injected could be determined. To determine the efficiency (the fraction of source fluid that goes through the sink), titration method was used using HCL-NaOH as acid-base combination and Phenolphthalein as pH indicator. Flow visualization results and the efficiency values show that (a) the sink flow rate required for 100% efficiency increases if the horizontal separation between source and sink will increase for both sink diameters, (b) the sink flow rate required for 100% efficiency is lower for the case of 20 mm sink diameter than that of 10 mm sink diameter keeping the horizontal separation constant, (c) there is increase in the sink flow rate required for 100% efficiency with increase in the vertical separation between the source and the sink and (d) if the source and the sink are at 90^0 , then the sink flow rate required will be more compared to that of configuration with 0^0.

T-2B-2. ANALYSIS OF FLOW CHARACTERISTICS AROUND CROSS FLOW FAN OF ROOM AIR CONDITIONER USING VISUALIZATION TECHNIQUE

S. H. LEE, *Sungkyunkwan University, Korea*, S. U. NA, *Samsung Electronics, Korea*, G. KANG, *Samsung Electronics, Korea,* H. S. KO, *Sungkyunkwan University, Korea,* Whole flow fields of a room air conditioner (RAC) have been visualized by a Particle Image Velocimetry (PIV) technique to analyze the flow structure by various inlet and outlet angles, and to control an eccentric vortex which affects an efficiency and noise of the RAC. A test model with 5 stages of a cross flow fan has been manufactured and a transparent acryl has been installed at the side of the test model for the PIV experiment. The inlet and outlet flows and the flow inside the cross flow fan have been analyzed by varying the inlet grill angles and outlet blade angles. The movement of the eccentric vortex has been investigated experimentally by developing the measurement technique for the inner flow field of the cross flow fan, and the relationship between the control of the eccentric vortex and the inlet and outlet angles has been investigated in this study.

T-2B-3. UNSTEADY MEASUREMENTS OF TURBULENT BOUNDARY LAYER FLOW PAST A 2D SQUARE CYLINDER BY USING TR-PIV

L. L. SHI, J. J. WAN, Y. Z. LIU, *School of Mech. Eng., Shanghai Jiao Tong University, China,* The turbulent boundary layer flow past a 2D square cylinder flush-mounted on a plane wall was extensively measured in a view to comprehensively understand the unsteady behaviors buried in the separated and reattaching flow. A TR-PIV setup was established by integrating a 1.8W semiconductor continuous laser and a high-speed camera (APX, Japan). A low-speed recirculation water channel was constructed by acrylic 10mm in thickness; the water was circulated by an electro-magnetic pump (Iwaki, Japan). The cross-sectional size of the working section is 150mm in width and 200mm in height. A square cylinder 15mm in width and height was flush mounted on a false plate 30mm above the bottom wall of the water channel, spanning the width of the working section. The free-stream velocity of the water flow was 0.15m/s. A polymer particles with averaged diameter 10um was seeded into the water fluid, which were illuminated by a laser sheet 1mm in thickness (λ = 532nm). Distribution of the seeding particles at the region downstream of the cylinder was captured at the rate of 500Hz. Statistical characteristics of the turbulent separated and reattaching flow were analyzed in terms of time-averaged velocity field, turbulence intensity field, velocity spectrum and cross-spectrum of velocity fluctuation and distribution of the reverse-flow intermittency. Unsteady behaviors of the vortical structures buried in the separated and reattaching flow were obtained from auto- and cross-correlation of velocity fluctuations, and the conditionally-averaged velocity field. The POD analysis of the vortical structures was given in the present study.

T-2B-4. VOLUMETRIC 3 - COMPONENT MEASUREMENTS USING V3V – AN INNOVATIVE APPROACH TO INSTANTANEOUS 3D FLOW MEASUREMENT

Rajan MENON, *TSI Incorporated, USA,* Dan TROOLIN, *TSI Incorporated, USA,* Solving flow problems in industry and research requires quantitative information of the 3-dimensional instantaneous velocity vector field. Flow structures, vorticity fields and other flow properties and their spatial and temporal variation are extracted from the instantaneous velocity vector fields in a volumetric region. The new Volumetric 3-Component Velocimetry (V3V) system, described here, uses a unique image capture system with three apertures to measure the instantaneous flow velocities at thousands of points in a volume. Unlike other imaging techniques where multiple cameras are individually mounted on a frame, V3V system uses an integrated imaging system that does not need any focusing, aperture and/or tilt adjustment. And, the axis of the measuring region and the standoff distance for the V3V Camera probe are predefined. The measuring region size, in the shape of a cube, was 140 x 140 x 100 mm and the typical stand-off distance was 500 mm. To measure the velocity field in a volume, a volume illumination system using a pulsed YAG laser is employed. Images captured simultaneously through the three apertures of the V3V camera are combined and processed to get the three-dimensional location and displacements of the particles. The HYPERSTREAMING system provides