

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° , then the sink flow rate required will be more compared to that of configuration with 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 acrylic 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 ($\lambda = 532\text{nm}$). 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 standoff 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

image capture and data transfer to the computer at the full frame rate of the V3V camera. *INSIGHT* V3V software package provides control and operation of the V3V system and particle image processing to get flow properties and graphical displays. The V3V system was used to measure flow induced by vortex motion, flow behind a sphere, and flow created by moving or flapping surfaces. The flow properties including the temporal and spatial development of the flow field provide new, unique information to people involved in system optimization, product design, and turbulence research.

14:30 ~ 15:50 (Room 103)

Vortex Flows

Session Chair : Prof. S. Izawa, Tohoku Univ/Japan

T-2C-1. THE VORTEX EVOLUTION AND MIXING CHARACTERISTICS OF A PASSIVE SCALAR IN A COAXIAL JET FLOW

H. S. CHOI, *Korea Institute of Machinery and Materials, Korea*, T. S. PARK, *KyungPook National University, Korea*, In the present study, the flow and mixing fields of a free coaxial jet is investigated by using computational fluid dynamics. For the numerical study, unsteady finite volume method is used with modified PISO algorithm and higher-order spatial discretization schemes, i.e. the forth-order COMPACT and central differencing schemes. In the present study, the coaxial jet flow consists of a central circular jet and an annular jet which surrounds the central jet. Especially, the ambient flow around the coaxial jet flows moves parallel with the coaxial jet flows. In the present study, velocity ratio between the central jet flow and the annular jet flow is changed as well as the velocity ratio between the annular jet flow and the ambient flow. The development of vortices generated from coaxial jet flow and the scalar field affected by the vortices are scrutinized numerically. With increasing the velocity ratio between the central and the annular jet flows, the peculiar vortex evolution pattern develops between them. In some cases of the present study, the vortices show a general vortex evolution pattern such as roll-up, pairing between two vortices and vortex break-down process. Firstly, in smaller velocity ratio, vortices originated from the inner and outer layer of the annular jet flow dominate the whole flow fields. With increasing the velocity ratio the vortices from the central jet flow are able to maintain their identity and can develop further downstream. And these vortices interact vigorously with the vortices developed from the outer layer of the annular jets with increasing the velocity ratio. Finally, the vortices from the central jet dominate the vortex evolution of the entire flow fields in the highest velocity ratio case. These vortex evolution patterns greatly affect the mixing fields and the passive scalar fields follow well with the flow fields in the present study. The global mixing efficiency greatly depends on strong engulfment or entrainment of the fluid parcels caused by vortex evolution process. So, these entrainment or engulfment motions are changed by varying the ratio of three different flows, i.e. the central circular jet, the annular jet and the ambient flows. Especially, with increasing velocity ratio between the annular jet and the ambient flow, the shear layer between them becomes weak with faster ambient flow case. In this case, the vortex evolution is suppressed and also the mixing rate is decreased.

T-2C-2. UNSTEADY FLUID FORCES AND VORTICAL STRUCTURE ACTING ON A THREE-DIMENSIONAL AIRFOIL

H. HASEGAWA, *Akita University, Japan*, A. NAKAMURA, *Akita University, Japan*, K. TANAKA, *Akita University, Japan*, Many studies on unsteady fluid forces have been carried out with the numerical and experimental approaches in order to understand the generating mechanism of unsteady fluid forces. In the present study, unsteady fluid forces acting on a discoid airfoil in the sinusoidal pitching motion were measured by using a ring structure 2 elements balance. Furthermore, the vortical fields in the wake of the airfoil were also investigated by using an X-type hot wire probe that was supported by a three-axis computer-controlled unit. The NACA0015 profile airfoil was used as the test model. Experiments were performed in a low speed wind tunnel. Tunnel speed was set to simulate the freestream Reynolds number of 1.0×10^5 ($Re=1.0 \times 10^5$) based on airfoil chord. Dynamic lift and drag coefficients were compared to the steady ones. For the airfoil in pitching motion on the static stall angle of attack, the delay of stall was observed and the maximum lift coefficient was greater than the maximum lift coefficient under stationary conditions. Degree of hysteresis in dynamic lift curve is found to be a strong function of reduced frequency. The strength of the counter-rotating vortex pair located downstream of the airfoil increased with increasing the lift force. The counter-rotating vortex pair collapsed in shape and the vortices elongated along the trailing edge profile of the airfoil at the instant of stall. For the pitching center over the

static stall angle, the stall delay and the increment in maximum lift coefficient were not observed at high pitching frequencies. The unsteady characteristics of a discoid airfoil were affected by an angle of pitching center and the pitching frequency.

T-2C-3. EFFECTS OF WHITCOMB'S WINGLET ON THE WING-TIP VORTICES

M. H. SOHN, *Korea Air Force Academy, Korea*, The present study investigates the wing-tip vortex of a three-dimensional wing for two different wing-tip configurations by using a smoke-wire visualization and PIV measurement of the near flow-field of the wing. The planform, profile, and cross-sectional views of the wing-tip vortices of a half-wing model with an aspect ratio 3.2 and different wing-tip configurations, simple fairing and Whitcomb's full winglet, were visualized at various angles of attack. The distinguishing characteristics of the vortex formation and structure for different wing-tip configurations that were revealed by the visualization results were also backed by a quantitative PIV measurement of the wing-tip trailing region. The wing model used in the present study is made of AA 5052 aluminum and has a taper ratio of 0.5 as well as different section geometry: its root section is a NACA 63₂-215 while its tip-section is a NACA 63₁-212. The wing model was twisted with a twist axis at the 30% chord line and with a washout angle of 2°, and is of basic configuration having no control surface, high-lift devices, and fuselage parts. The tip-region of the wing model was modified by attaching a simple fairing and a full winglet to the basic square-cut wing-tip. The upper part of the winglet has a root chord length of $0.7C_{tip}$ where C_{tip} is the wing-tip chord (128.5mm), and a span of $1.0C_{tip}$, as well as a taper ratio of 0.5, a sweepback angle of 20 deg, and a cant angle of 15 deg. The lower part of the winglet has a root chord length of $0.4C_{tip}$, and a span of $0.23C_{tip}$, as well as a taper ratio of 0.4, a sweepback angle of 52 deg, and a cant angle of 36 deg. The smoke-generating oil for the visualization was a mixture ratio of 93% paraffin oil and 7% lubricating oil, which produced smoke streak-lines of appropriate smoke density and discreteness. Teikoku alloy wire of diameter 0.25mm was used as smoke-generating wires. The free stream velocity of the visualization was set to 3.6m/sec. The Reynolds number based on the chord of the mid-span section was about 5.5×10^4 . PIV System used consists of a double-pulse Nd:YAG laser (Vlite-200) with a maximum pulse energy of 2x200 mJ at a repetition rate of 10Hz, a 8-bit digital CCD with a resolution of 2048x2048 pixels, and a PC equipped with DaVis FlowMaster software and a synchronization board developed by LaVision GmbH. An aerosol Generator was used for DEHS(C₂₆H₅₀O₄) particle seeding. The PIV data in this study was an ensemble average of 20 instantaneous velocity fields. The free-stream velocity of the PIV measurement was 26.0m/sec, which corresponds to the Reynolds number of 4.12×10^5 . The visualization pictures clearly captured the characteristics of the wing-tip vortex formation and structure. The comparison of the two different wing-tip configurations showed that the wing-tip vortices of Whitcomb's winglet configuration were reduced in strength, and displaced outboard and upward, at least in the near-wake region, which resulted in an increased lift-to-drag ratio for the Whitcomb's winglet configuration.

T-2C-4. VORTEX RING RELATED TO THE MOTION OF A BODY

S. ICHIKAWA, *Toyo University, Japan*, O. MOCHIZUKI, *Toyo University, Japan*, The purpose of this study is to understand the relation between a vortex ring shed from a jellyfish and thrust force of the motion of a jellyfish. The starting vortex ring was generated by a snappy motion of the tip of the skirt of a jellyfish. It looks like a starting vortex at a trailing edge of an airfoil moving suddenly in two-dimensional flow. The lift force of the airfoil can be calculated by the Kutta-Joukowski theorem. In contrast to a two-dimensional ideal flow, is there a relation between the circulation of the vortex ring and a three-dimensional body moving? This question came from the result of the observation of a jellyfish swimming. The jellyfish did not get thrust from ejection due to contraction of the skirt in our experiment. The thrust appeared when a vortex ring was generated at the tip of the skirt. Therefore, we think that a vortex ring is attributed to the thrust generation of the jellyfish. The translating velocity of the vortex ring itself is related to its circulation, but the relation between the force acting on the body and the circulation of the vortex ring generated by the motion of the body is not clear. To understand the relation, the flow field induced by a circular disk moved by the known force was investigated experimentally in this study. We measured the velocity vector field around the disk by using a PIV method to estimate circulation of the starting vortex ring. It was found that the circulation increased with the square root of the force which is necessary to move the disk. This showed the same relation of the energy of a vortex ring and its circulation which was presented by Saffman.