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 pair of laser pulses was generated using an AOM chopper. The hologram fringe images were captured using a highspeed digital CMOS camera (Photron, FASTCAM Ultima-APX) with a spatial resolution of 1024×1024 pixels (17μm/pixel). Water was passed the curved micro-tubes with inner diameters of 100μm and 300μm. The micro-tubes were made of FEP, which has a refractive index of 1.338, similar to that of water. To reveal the flow characteristics at high Dean numbers, the trajectories of fluid particles were evaluated experimentally from the whole 3D velocity field data measured using the HPTV technique. The initial location for the fluid particle trajectories was taken at the radius r/f=-0.4 in the given cross-section of the tube. Here, N denotes the number of pitches (from the initial cross-section) required to reach the crosssection at which the trajectory was extracted.

T-1B-3. ADV MEASUREMENT OF FLOW IN SEDIMENTATION BASINS

B. FIROOZABADI, H. JAMSHIDNIA and A. HOUSHMAND, *Center of Excellence in Energy Conversion, Mech. Dept., Sharif Univ. of Tech., Iran,* Flow field in a sedimentation basin plays a very important role in a way that a sedimentation tank must be conducive to sedimentation. In fact, hydraulics of sedimentation tank is one of the great important factors that influence removal efficiency. There have been some investigations on flow field in the literature but there is relatively few detailed experimental measurement of velocity field. Since by having a complete understanding of hydraulics of sedimentation tanks it is possible to find new solutions to modify their flow field to achieve better performance of these facilities, in this paper the flow field in a rectangular sedimentation basin is measured using ADV at different concentrations. Furthermore the applicability of ADV to measure flow field by measuring the velocities point-wise is shown. The rectangular channel of height 0.4 m had a working area of length L=8 m long, width of W=0.2 m (Fig.1). Data are available at an output rate of 25 Hz. The 3-D velocity range is 2.5 m/s, and the velocity output has no zero-offset. Structure of the flow has been studied by investigating velocity profiles at various sections along the channel and comparing them together. In addition, the structure of the flow has been studied at neutral flow condition (C_{inlet}=0) and particle laden flow at two low and high inlet concentrations. Analyzing the measured data revealed that in neutral flow that there is no particle, the hydrodynamic flow pattern is almost uniform along the tank at the main part of depth, except near the inlet. On the other hand, in particle laden flow sediment driven density current exists which causes the usual pattern of flow to deviate to a great amount from uniformity. In Fig.1 this difference according to the achieved data is presented. C in this figure represents the inlet concentration. Additionally, investigation of the effect of concentration on the hydrodynamic of flow pattern revealed that at higher concentration a bottom current with higher maximum velocity near the bed is induced. In this case the bottom current is strong enough that causes a return surface flow at the upper parts of the channel. The achieved data not only represents the applicability of ADV for measuring flow fields of sedimentation channels but also gives quantitative results revealing more accurate information on the structure of flow. Perhaps by considering these results it would be possible to find solutions such as installing a baffle to improve the sedimentation performance.

T-1B-4. EXPERIMENTAL AND COMPUTATIONAL STUDIES ON A CENTRIFUGAL SEPARATOR FLOW OF GAS AND LIQUID

S. P. NAGDEWE, H. D. KIM, *Andong National University, Korea*, D. S. KIM, A. SURYAN, *FMTRC Daejoo Machinery Co. Ltd., Korea,* A gas liquid centrifugal separator is widely used in industry on account of its simple geometry and little maintenance. These separators have considerable advantages over filters, scrubbers or precipitators in term of compact design, lover pressure drop and higher capacity. A gas liquid centrifugal separator is a device that utilizes centrifugal forces and low pressure caused by rotational motion to separate liquid from gas by density differences. Efficient and reliable separation is required for the optimum operation of separators. These separators are often operated at less than peak efficiency due to the entrainment of separated liquid through an outlet pipe which is closely associated with the very complicated flow phenomena. Design parameters such as length of separation space, vane exit angle, inlet to outlet diameter ratio, models for separation efficiency and pressure drop as a function of physical dimension are not available in literature. This gives designer very little scope for available data. The aim of present study is to perform a computational study to get higher efficiency for gas liquid separators. A computational study has been carried out with the help of CFD tools to analyze a separation performance of a centrifugal separator. The computational results are compared with experimental results for their validity. The best design parameters are analyzed based upon obtained results, tangential velocities, vortices, total pressure losses. From the present study several attempts are made to improve the performance of conventional centrifugal separators.

Viscous Flows

10:30 ~ 11:50 (Room 103)

Session Chair : Prof. J. Sung, SNUT/Korea

T-1C-1. DYNAMICS OF ACCELERATED CURVED VISCOUS FLOWS

Ajay Vikram SINGH, A. KUSHARI, *Department of Aerospace Engineering, IIT Kanpur, India,* The present work deals with the dynamics of accelerated curved flows. The results of the study are important in thrust vectoring used in modern military aircrafts. So far thrust vectoring has been proved to be a boon if we look from the flight mechanics point of view. But the fluid mechanics of such accelerated curved flows is quite complex. This paper deals with the identification and analysis of effective flow angle in curved accelerated viscous flows, which is generally different from the geometric angle of curvature. The dynamics of accelerated curved flows has been analyzed in detail to understand the underlying physics responsible for the divergence of the effective flow angle from the geometric curvature. Thrust vectoring is the ability of an aircraft or other vehicle to direct the thrust from its main engine(s) in a direction other than parallel to the vehicle's longitudinal axis. The technique was originally envisaged to provide upward vertical thrust as a means to give aircraft vertical (VTOL) or short (STOL) takeoff and landing ability. Subsequently, it was realized that using vectored thrust in combat situations enabled aircraft to perform various maneuvers not available to conventional-engine planes. Jet deflection to obtain forces for enhancing aircraft performances is the aim of thrust vectoring (TV) technology. Dynamics of accelerated curved flows carry a great importance in thrust vectoring nozzles as the exit jet gets deflected by significant angles.

The present work deals with the fluid mechanics and dynamics of accelerated curved flows involving both numerical and experimental studies. An attempt has been made to understand the physics of such accelerated curved flows, particularly the effect of geometric curvature and the flow Reynolds number on the effective flow angle. The dynamics of accelerated curved flows is also important in case of curved conduits like A.C. ducts and the flow through turbine blade passages.

T-1C-2. VORICAL STRUCTURES FROM CONTROLLED CIRCULAR JET

Dae Il LEE, *Seoul National University, Korea*, Jungwoo KIM, *University of Florida, USA*, Haecheon CHOI, *Seoul National University, Korea,* The control of jet has been an important issue in engineering applications such as the noise reduction, mixing enhancement, and combustion-efficiency increase. Especially, controls based on the axial and/or helical excitations at the jet exit have been applied to modify the jet evolution into the bifurcating, trifurcating, and blooming jets. The bifurcating jet has been studied, but the trifurcating and blooming jets have not been investigated in detail. In the present study, we investigate various types of vortical structures from controlled circular jet including blooming and trifurcating jets. Large eddy simulations are carried out at $\mathrm{Re}_D = 4300$ based on the jet-exit velocity (U_J) and jet diameter (D) with a dynamic Smagorinsky model in the cylindrical coordinate system. The number of grid points is $449 \times 144 \times 129$ in the axial, radial, and azimuthal directions, respectively. For the jet inflow, a top-hat velocity profile with a laminar Blasius profile near the wall is used, together with background disturbances. The jet inflow condition is given in the below:

$$\frac{u_{x=0}(r,t)}{u_{x=0,uc}(r)} = \left[1 + A_a \sin(2\pi St_a t) + A_h \sin(2\pi St_h t + \gamma)(\frac{2r}{D})\right]$$

, where St_a and St_h are the Strouhal numbers based on the axial and helical forcing frequencies, respectively, A_a and A_h are their amplitudes, and γ is the relative phase between two excitations. The subscript *uc* denotes the case of uncontrolled jet. The peak point associated with St_h is defined as a point having the maximum velocity at the jet exit during one cycle of axial excitation, and the peak angle (α) is the azimuthal angle between the adjacent peak points. The peak point determines the unique route of each discrete vortex ring, so that the characteristics of jet evolution are explained in terms of the peak point and angle. According to the peak angle, the number of branches varies from 1 to 5. The curvature of branches increases with decreasing peak angle at a given number of branches.

T-1C-3. STOKES EXPANSION FOR LAMINAR FLOW THROUGH