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 $Re_D = 4300$ 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 $\gamma$ 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 ($\alpha$) 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**