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.

### 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

**A SLOWLY ROTATING STRAIGHT PIPE: EFFECT OF ROTATING RATIO**

Kamyar MANSOUR, *Department Of Aerospace Engineering And New Technologies Research Center Amir Kabir University of Technology Tehran, Iran,* We consider fully developed steady laminar flow through a pipe that is rotating slowly about a line perpendicular to its own axis .The solution is expanded by computer in powers of a single combined similarity parameter introduced by [1] $K = RR_r$, keep $d = \frac{R_r}{R}$ the rotating ratio fixed where R is a Reynolds number based on axial velocity $W_0$ and $R_r$ is the Reynolds number based on rotational velocity (a$\Omega$). . Then the series extended by means of symbolic calculation up to 18 terms. Analysis of these expansions allows the exact computation for arbitrarily accuracy up to 50000 figures. Although the range of exactness is almost the same order of the radius of convergence but Pade approximation lead our result to be good even for higher values of both parameters K and rotating ratio $d = \frac{R_r}{R}$

**T-1C-4. INSTABILITY MODES OBSERVED IN NATURAL TRANSITION OF AN AXISYMMETRIC WAKE**

S. HOSHINO, A. INASAWA and M. ASAI , *Department of Aerospace Engineering, Tokyo Metropolitan University, Japan*, Y. KONISHI, S. TAKAGI and H. SAWADA, *Institute of Aerospace Technology, Japan Aerospace Exploration Agency, Japan,* The linear instability of axisymmetric laminar wake of a body of revolution with NACA0018 airfoil cross-section is investigated under natural disturbance conditions experimentally. The experiment is carried out in a low-turbulence wind tunnel with a square test section. An axisymmetric body is suspended by the Magnetic Suspension and Balance Systems (MSBS) to avoid undesirable influences of mechanical supports on the disturbance development. In the MSBS, the model with embedded permanent magnet is kept staying at the same position in the flow by well-controlled magnetic force. The experiment is conducted at three Reynolds numbers (based on the free-stream velocity and the maximum diameter) Re = $1.4 \times 10^4$, $1.9 \times 10^4$ and $2.4 \times 10^4$, and the detailed measurements are conducted at Re = $1.9 \times 10^4$. Measurements of time-mean velocity and velocity fluctuations are done by using hot-wire anemometers. In addition to a single I-type hot-wire probe, multi-hot-wire probe which consists of six I-type sensors arranged in the azimuthal direction with equal angle of 60° is used to identify helical instability modes. Stability calculations based on the inviscid linear stability equations are also done to understand the experimental results. For this axisymmeteric model, the flow is slightly reversed in the region close to the trailing edge. The reversed flow velocity is at most 3 % of the free-stream velocity. Unlike two-dimensional wakes, the instability nature of such an axisymmetric wake does not exhibit the nature of absolute instability. That is, disturbances grow exponentially with the streamwise distance depending on the frequency. The stability analyses of the velocity profiles measured in the reversed flow region also confirm the experimental observation. Spatially-growing disturbances are found to be helical wave modes with azimuthal wavenumber of 1 as predicted by the linear stability theory for the axisymmetric wake. The most amplified frequency is close to that calculated from the linear stability theory.

| 10:30 ~ 12:10 (Room 104) |
| --- |
| **Rotating Flows** <br> Session Chair : Prof. Q. H. Nagpurwala, MSRSAS/India |

**T-1D-1. EFFECTS OF BLADE PROFILE AND NON-UNIFORM TIP CLEARANCE ON THE PERFORMANCE OF WELLS TURBINE**

M. TAKAO, *Matsue National College of Technology, Japan*, S. NAGATA, *Saga University, Japan*, K. TOYOTA, *Saga University*, *Japan*, M. KIDO, *Saga University, Japan*, T. SETOGUCHI, *Saga University, Japan,* Wells turbine is a self-rectifying air turbine which is expected to be widely used in wave energy devices with oscillating water column (OWC). There are many reports which describe the performance of Wells turbine. However, Wells turbine has inherent disadvantages: lower efficiency, poorer starting and higher noise level in comparison with conventional turbines. In order to enhance the performance of Wells turbine, some rotor blade profiles have been recommended by various researchers. According to previous studies, a symmetrical airfoil of NACA four digit series is a preferable one when the turbine is operated at low Reynolds number. Especially it has been shown that the NACA four digit series with thickness ratio of approximately 20% is a recommended one for the rotor blade. However, the stall angle of Wells turbine with this blade profile is not so high. The aim of this study is to investigate the effect of rotor blade profile on the performance of Wells turbine. The experimental investigations have been performed by use of

test section with a casing diameter of 300mm. The tested turbine has 6 blades with a chord length of 90mm. In the study, four kinds of blade profile were selected and tested by model testing under steady flow condition. The types of blade profile are as follows: NACA0020; NACA0015; modified NACA0015 and modified Eppler472. Further, the effect of non-uniform tip clearance on the turbine performance was investigated under steady flow condition and the results were compared with those of Wells turbine with uniform tip clearance. The uniform tip clearances are 0.5mm and 1.0mm. The clearance in the case of non-uniform type increases gradually with chordwise, and the clearances at leading and trailing edges are 0.5mm and 1.0mm, respectively. As a result, it seems that a suitable choice of these design factors is blade profile of modified Eppler472 and non-uniform tip clearance in the study.

**T-1D-2. FLOW AND DRAG CHARACTERISTICS ON A ROTATING SURFACE WITH RADIALLY MODULATED TOPOGRAPHY**

M. S. YOON, *Korea Testing Laboratory, Korea*, J. S. PARK, *Halla University, Korea*, J. M. HYUN, *KAIST, Korea,* Flow in a rotating cylinder driven by the differential rotation has been studied, when the bottom end-wall disk has a sinusoidal surface roughness, i.e., $z_b = a \cdot \cos(2\pi Nr)$ where $z_b$ denotes the local height of bottom disk surface, $a$ the wave amplitude and $N$ the wave number of the bottom surface. To parametric studies on various surface topography, the 40-cases of numerical computation, as varying both of wave amplitude and wave number, were performed for an incompressible steady Navier-Stokes equation. The system Reynolds number is assumed to be large, and then a boundary layer flow pattern prevails. From the linear analysis by Park, Yoon and Hyun(2008), it showed under the assumption of mild slope, i.e., $|z_b'| < O(1)$ that flow characteristics on three velocity components (u, v, w) and drag on the rotating surface could be correlated with a single parameter $(aN)$, not correlated independently with two parameters of wave amplitude $a$ and wave frequency $N$ . To test whether previous linear results are still available or not in the case of nonlinear flow by rotating wavy disk , full nonlinear numerical analysis to the Navier-Stokes equations and linear regression analysis to obtained data have performed and a comprehensive description is given on roughened surface effect of the disk to the flow fields. Dynamical ingredients of surface roughness on nonlinear flow field are qualitatively consistent with linear case, i.e., all flow characteristics being correlated with single roughness parameter ( $aN$ ). However, parametric dependence shows some deviation such as all flow variables depend on $(aN)^2$ for linear case and $(aN)^1$ for full nonlinear case. Numerical solutions were acquired by utilizing the well-established SIMPLER algorithm [Patankar, 1980]. In order to deal with the sinusoidal wave surface geometry, transformations were made to introduce the body-fitted coordinates in the computational domain. Iteration was declared converged when the relative changes of the flow variables were less than $10^{-5}$ between two successive iteration levels. Extensive trial-and-error tests were performed for the computational grid network. The (170*130) grid system was adopted for the computation. in order to study wavy bottom disk, we also tried at the same parametric values $R_e = 3.0 \times 10^3$ (Reynolds number) and $A_r = 1.0$ (Aspect ratio).

**T-1D-3. AN EXPERIMENTAL STUDY ON THE ASPECT-RATIO EFFECT OF A CROSS-FLOW IMPELLER**

J. FUNAKI, *Department of Mechanical Engineering, Doshisha University, Japan*, Y. ONISHI, Y. IIDA, *Graduate School, Doshisha University, Japan*, A. TAKUSHIMA, *Samsung Yokohama Research Institute, Japan*, K. HIRATA, *Department of Mechanical Engineering, Doshisha University, Japan*, According to Eck (1973), the cross-flow impeller, or the cross-flow fan, was invented by Mortier in 1892, and it had been used as a fan for ventilation of mines in those days. Recently, because the cross-flow fan, or the cross-flow impeller, can easily generate almost uniform, two-dimensional, thin and wide flow, in the direction perpendicular to its axis, the impellers have been widely used for industrial equipments and home electric appliances. The purpose of this experimental research is to investigate the aspect-ratio effect on the flow around and inside cross-flow impellers. Measurements of the velocity distributions were made by PIV in order to investigate the characteristics of the eccentric vortex. The cross-flow impeller, which consists of transparent acrylics, rotated in a stationary fluid without any casing. The working fluid was air, and minute particles of olive oil were used as the tracers for flow visualization. The velocity vectors by PIV of the forward-cambered-blade impeller, show that the eccentric vortex revolves steadily at a constant speed. As the rates of the high-speed video camera are much higher than the impeller's rotation speed, we can