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 = 1.4×10^4, 1.9×10^4 and 2.4×10^4, and the detailed measurements are conducted at Re = 1.9×10^4. 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
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

observe the inside flow as well as the outside flow at an arbitrary instant. The flows shown in the figures are on the mid-span plane. Here, all velocities are normalized as u/u_2. The maximum value of u/u_2 is about 1 at Re=2500. As a result, the flow can observe the eccentric-vortex revolution by using hot-wire measurements and flow visualizations. By HWV measurements and by conventional flow visualizations, the authors divide the flow around the impeller into three areas, and observe the eccentric-vortex revolution inside the impeller. Using a PIV tequnique, the authors quantitatively show velocity distributions, where accuracy is checked in comparison with HWV results. The aspect ratio effect on the outflow-rate coefficient C_{Q_0} is not negligible at aspect ratios less than one.

T-1D-4. FLOW PREDICTIONS IN TRANSONIC COMPRESSOR ROTOR

V. R. KALAMKAR, G. R. SHEVARE and B. ROY, *Department of Aerospace Engineering, IIT, Bombay, India,* The simulation of complete flow field in a transonic axial compressor rotor represents a considerable challenge for flow prediction methods. The current paper reports on the study of complete flow physics and secondly applicability and limits of the steady state model for transonic compressor rotor. An in-house code is developed and applied to study this problem. . A steady state analysis of the transonic compressor rotor is carried out by using three dimensional finite volume based explicit RANS solver using structured multi-block meshes. The present investigation uses numerical scheme as AUSM. The code runs on parallel architecture by using MPI libraries. The effect of turbulence is taken care by using Spalart Allmaras model. The current paper reports investigations aimed at advancing the understanding of the flow field near the casing of a transonic compressor rotor. The role of tip clearance flow and its interaction with the passage shock at peak efficiency and near stall operating condition is analyzed. The numerical results such as the compressor performance maps as well as spanwise distribution of total pressure and total temperature have been first compared with the experimental data for design speed and for tip clearance of 0.356 mm. Then, the numerically obtained flow fields are interrogated to identify the roles of flow interactions between the tip clearance flow, the passage shock and the blade/end wall boundary layers. Mostly, study is carried out for the analysis of radial extension of shock/vortex interaction by presenting Mach number distributions on cross channel planes at different axial stations such as 20%, 108% and 138% chord from leading edge.

10:30 ~ 11:50 (Room 105)
Free Surface Flows (IV)
Session Chair : Prof. H. Liu, Shanghai Jiao Tong Univ/China

T-1E-1. SOLVING THE PROBLEM OF WATER WAVES OVER A STRATIFIED MUDDY BED BY MATHCAD

C. O. NG, H. S. CHIU, *The University of Hong Kong, Hong Kong,* This work marks the debut of a Mathcad worksheet, which has been developed to aid engineers in the calculation of properties of a surface wave propagating in water over a two-layer viscoelastic muddy bed. In each bed layer, the mud is modeled as a viscoelastic Voigt medium with constant viscosity and shear modulus of elasticity. The worksheet is a very easy-to-use calculation tool. With the input of some basic parameters, such as the wave period and the fluid properties, one may get almost instantly the key results (wavenumber, wave damping rate, velocity components, pressure, and so on) upon the pressing of a key. The worksheet has been extensively tested by comparing with published results in the literature to ensure that it can produce reliable and accurate results. The worksheet is of value not only to practicing engineers, but also to researchers in the areas of coastal engineering, wave mechanics, and so on. The Mathcad worksheet file is available upon request. Those who are interested are welcome to contact the first author for a copy of the file.

T-1E-2. CURRENT MODELING OF WATER IN A RIVER MEANDER CONSIDERING CIVIL ENGINEERING PROBLEMS IN BUILDING THE COASTAL WALLS

S. M. MOSADDAD, *Islamic Azad University, Shooshtar branch, Iran*, A. A. BIDOKHTI, *Researching assistant of Geophysics Institute, Tehran university, Iran* and M. EZAM, *Ph.d of physical oceanography, IAU, Researches and sciences brach, Tehran, Iran,* River flows are not often straight and any small deviation in their paths can be enhanced due to the secondary flows. There are quite a few places in the south of Iran in which the rivers pass through cities which have lead to serious erosions. The example here is the Karoon River passing through the Shoushtar city,

Shotteit branch. there are numerous water structures which are often known as "Pole-Band" meaning bridge-dam that are used for water flow control as well as for crossing from one side to another side of river. Using the analysis of secondary flow in one bend of this river with and without crevasse we showed that the role of this old structure is substantial in reducing the secondary flow as well as flood control and also water resource management. The studied hydraulic structure called Band Mizzan is built at the meander of the Karoon River near the center of Shoushtar city as a crevasse channel about 1800 years ago (Sasanian Persian kingdom) that depth of the main channel is typically 4 m (±1 m inthe bed). It is as a result of existence of vorticity component normal to the main flow which goes through a curved path. So Drainage of bottom slow moving fluid into the channel, secondary circulation in the side channel can lead to deposition and blocking of the channel. We mainly find through this study that: Water division decreases discharge of water and coastal erosion would be decreased.

T-1E-3. NUMERICAL INVESTIGATION OF WAVE BEHAVIOR OF THERMAL DISTRIBUTION USING NON FOURIER CONDUCTION

E. IZADPANAH, *Yazd University, Iran*, S. TALEBI, *Yazd University, Iran*, M. MIRZAEI, *K. N. Toosi University, Iran,* M. H. HEKMAT, *K. N. Toosi University, Iran,* In this paper, temperature distribution in the flat plate and semi infinite plate has been investigated for three different boundary conditions (constant surface temperature, instant and constant heat flux) by means of Fourier and non-Fourier conduction equations. Governing equations have been solved by implicit finite difference method and a central difference scheme is employed in discretizing the spatial derivatives and a first and two orders in discretizing the time derivatives. In all problems, Non dimensional time step is taken as 0.00001 and non dimensional spatial step is considered 0.0001. The results show that in non-Fourier conduction, heat transfer has a wave movement. The formed thermal wave in the flat plate has a reciprocating movement. This movement repeats until the plate reach to thermal equilibrium. Also in initial times, the results of Fourier and non-Fourier conduction have difference with each other. Therefore, in problems which the time scale is small and the heat conduction to a body is rapid such as laser radiation to a body, applying the non-Fourier conduction is noticeable. The investigation of results shows that for a semi-infinite plate, Fourier conduction model at boundary condition of constant flux, can not predict temperature distribution in the initial times accurately. The results of Fourier model for heat flux distribution in semi-infinite plate at constant temperature state in initial times is incorrect. Totally, the Fourier equation is acceptable when the all three conditions exist:
 -The length scale is large.
 -The time scale is large.
 -The initial temperature of body is much grater than Kelvin zero.

T-1E-4. SIMULATION OF TSUNAMI WAVEGUIDING IN AN INDONESIAN COASTAL AREA

D. ADYTIA *LabMath-Indonesia*, A. SOPAHELUWAKAN *LabMath-Indonesia*, ANDONOWATI *LabMath-Indonesia & ITB Bandung Indonesia,* E. Van GROESEN *LabMath-Indonesia & UTwente Netherlands,* Field observations of effects of tsunamis on the coast show a high spatial variability that is not well captured by many numerical simulations. As one possible cause for this variability we introduced the phenomenon of Near-Coast Tsunami Waveguiding. Using a simple, synthetic bathymetry and geometry, we showed that the coastal approach over sloping bottom leads to largely enhanced wave amplifications by the presence of a narrow ridge that act as waveguide. In this paper we show that also in a realistic coastal area tsunami waveguiding can be convincingly observed in simulations.

10:30 ~ 11:50 (Room 106)
Drops and Bubbles (I)
Session Chair : Prof. Mohammad Ali, BUET/Bangladesh

T-1F-1. CAPILLARY INSTABILITY OF A CYLINDRICAL LIQUID COLUMN

Mohammad ALI, *BUET, Bangladesh,* A. UMEMURA, *Nagoya University, Japan,* This paper describes the formation of capillary wave and its instability during the contraction of a square cylindrical liquid column. The breakup behavior for the present configuration of the liquid column is investigated and found some significant differences from those predicted by conventional jet atomization theories. The formation of capillary wave is initiated by the surface tension on the sharp edge of the square end of the