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_{Qo} 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 bend). 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

cylinder and the propagation of the wave occurs due to the effect of surface tension force on the motion of the fluid. The investigation reveals that before disintegration of the liquid the capillary waves become unstable and the source of making the wave unstable is inherently developed by the system.

T-1F-2. SIMULATION OF DROP MOVEMENT ON AN INCLINED SURFACE

F. TAVAKOLI, *Isfahan University of Technology, Iran,* E. SHIRANI, *Isfahan University of Technology, Iran,* Many engineering and industry applications involve moving of droplets over solid surfaces. Numerical simulations of the dynamics of droplet impact and spreading on inclined surfaces are presented here. The simulation utilizes PLIC-VOF to model the interface. An improved surface tension method is used and contact angles around the droplet are applied as a boundary condition. The effects of the droplet velocity and surface inclined angle on the deformation and movement of the droplet are studied. The main problem in the numerical modeling of surface tension forces is the production of so-called "spurious" or "parasitic" currents. They are small but growing flows, which are generated due to the different density of two phases in the interfacial region. Spurious currents affect the interface shape and produce unphysical results. The more commonly used surface tension treatment methods are the CSF and the CSS. SGIP method was developed by Seifollahi, et al., EJM, 2008 which uses PLIC-VOF methods. In SGIP method the normal and the interface surface area needed for the calculation of the surface tension force are calculated more accurately. This method is applied to a staggered grid and it is referred to as Staggered Grid Interfere Pressure. First we consider a 1 cm stationary water drop in air at standard conditions and examine four different surface tension models which are CSS, CSF, SGIP and CSF-BKZ (the latter is proposed by Brackbill, Kothe and Zemach). With respect to the production of spurious currents and calculation of pressure jump it is found out that the SGIP model has the best performance and thus we used it for our other cases. Bussmann et al., Phys. of Fluids, 1999 successfully simulated the drop spreading on an inclined surface using dynamic contact angle. Afkhami and Bussmann, ILASS, 2006 extended the work of Bussmann et al. by discussing the effect of different implementations of the contact angle and the contact line velocity on the predictions of drop spreading. Based on experimental observations, finally they introduced a model for contact angle as a function of velocity contact line. However, such relationship is difficult to obtain for a general case. Using SGIP model and applying dynamic contact angle, the deformation and movement of a droplet on inclined surfaces is simulated. Simulations comprise impact of a 3mm droplet on an inclined surface, with inclination of 30, 45 and 60 degrees and two impact velocities of 0.5 and 1 m/sec. It shows that a droplet on a surface with small inclination spreads more rapidly and moves down more slowly. Additionally by increasing of droplet impact velocity, the spreading of the droplet will be increased.

T-1F-3. VELOCITY AND ACCELERATION MEASUREMENT OF SPHERICAL CAP BUBBLE USING PIV/LIF AND SHADOWGRAPHY

M. J. SATHE, *Department of Chemical Engineering, UICT, Mumbai, India,* I. H. THAKER, *TSI Inc, Bangalore, India*, T. E. STRAND, *TSI Inc, Shoreview, USA* and J. B. JOSHI, *Department of Chemical Engineering, UICT, Mumbai, India,* Simultaneous measurements of Bubble Size, Shape, Velocity and acceleration along with surrounding liquid velocity are presented for the case of a single spherical cap bubble rising in water. Although the current technique is restricted for the case of low gas hold-up, the information collected is valuable for obtaining the CFD modeling parameters like Drag force, Virtual mass force, Lift force etc. Particle Image Velocimetry (PIV) is carried out with fluorescent tracer particles, along with Shadowgraphy to deduce the bubble shape and size. Bubble velocity and acceleration are obtained by Particle Tracking Velocimetry (PTV) applied to processed bubble images. Details concerning the synchronization between the PIV cameras, Nd: YAG laser, bubble generator and high speed camera are presented. Because of the versatility of the technique the same hardware can be used for different resolutions of the flow field, from few hundred microns to few centimeters. The PIV/PTV results for the spherical cap bubble are presented. Although scarce experimental data regarding flow field around cap bubbles in turbulent regime exist, the results are encouraging.

T-1F-4. AN APPLICATION ANALYSIS OF BUBBLE PUMP SOLAR WATER HEATER

LI Xuesong, *Gyeongsang National University, Korea*, GiTae PARK, *Gyeongsang National University,* Hanshik CHUNG, *Gyeongsang National University, Korea*, Hyomin JEONG, *Gyeongsang National University, Korea,* The experiment in this paper is based on bubble pump applied on solar water heating system. The equipment consists of the bubble pump, heater and heat exchanger. The complete system was instrumented to measure pressures, temperatures and flow-rates at various locations. For solar heating of domestic hot water, two common system types are thermosyphon and pumped. In the thermosyphon system, a storage tank is placed above the collector. As the water in the collector is heated, it will rise and naturally start to circulate around the tank. This draws in colder water from the bottom of the tank. This system is self-regulating and requires no moving parts or external energy, so is very attractive. Its main drawback is the need for the tank to be placed at a level higher than the collector, which may prove to be physically difficult. A pumped system uses a pump to circulate the water, so the tank can be positioned independently of the collector location. This system requires external energy to run the pump. It also requires control electronics to measure the temperature gradient across the collector and modulate the pump accordinglyAs a bubble pump air-pumped pump, we can use the mobile water bubbles up from the lower elevated to a height, so as to achieve the purpose of upgrading cycle pressure. Therefore, the bubble pump can be applied to split-type solar water heating system to replace the electric circulating pump, in the application of solar energy only at the same time to achieve the purpose of automatic cycle.

10:30 ~ 11:50 (Room 107-108)

Computational Fluid Dynamics (IV)

Session Chair : Dr. R. Kidambi, NAL Bangalore/India

T-1G-1. NUMERICAL SIMULATION OF CAVITY OVER HYDROFOIL BY USING BOUNDARY ELEMENT METHOD BASED ON POTENTIAL FLOW

A. R. MOSTOFIZADEH, *Malek Ashtar University of Technology, Iran*, M. PASANDIDEHFARD, *Ferdosi University, Iran,* S. GHOLIZADEH, *Malek Ashtar University of Technology, Iran,* In this paper numerical study of cavity over hydrofoils is considered by using boundary element method based on potential flow. For this purpose hydrofoil and cavity surface are approximated by panels. Then sources and doublets are distributed over these surfaces. In this method the length of cavity is assumed constant. A set of equations are obtained by applying boundary conditions over the hydrofoil and cavity surface with closing cavity condition, which they are solved together. An important advantage of this method is getting the answer in a short period of time along with low cost computations. Also, there is a good agreement between numerical and experimental results that shows the accuracy of this method.

T-1G-2. COMPARISON OF PERFORMANCE OF PPM WITH ROTATION AND KENICS STATIC MIXER

J. Y. C. LEONG, *Monash University, Malaysia*, C. F. THAN, *Monash University, Malaysia*, Y. W. OOI, *Monash University, Malaysia,* A comparison of the performance of a Kenics static mixer and a Partitioned-pipe Mixer (PPM) has been investigated. The Kenics static mixer is a commercially available mixer that incorporates the use of stationary helical mixing elements to direct the flow to promote mixing. The PPM, conversely, partitions the pipe into two semicircular ducts and introduces a rotation at the wall to mimic the effect of the helical shaped mixing elements available in the Kenics static mixer. The study of the PPM is of interest because it is able to capture the main features of the Kenics static mixer to a certain degree while maintaining a fairly simple geometry. However, most analytical work done in defining the flow fields within the PPM has been restricted to highly viscous creeping flows or Stokes flow. This study was conducted with the help of the commercially available Computational Fluid Dynamics software package, Fluent, to simulate the flow fields in both mixers. The use of numerical methods is attractive because it does not suffer from the restrictions of the analytical models and can therefore offer greater insight into the subtleties in both flow fields. The performance of both mixers will be evaluated primarily by its ability to mix two immiscible liquids, namely, palm oil triglyceride and methanol where the mixing of these two fluids is of interest in the palm oil transesterification process. The PPM and Kenics static mixer were compared in terms of pressure drop across the mixer as well as efficiency in performing the mixing. The simulated data showed that the simple PPM model was able to give insight into the performance of the Kenics static mixer within the range of Reynolds number studied. For further study, detail comparison of PPM and Kenics static mixer should include varying pitch and size of the mixer elements and wall rotation speed.