N. DIZADJI, *Islamic Azad University Science & Research Campus, Iran*, S. KANANPANAH, *Tehran University, Iran,* H. ABOLGHASEMI, *Tehran University, Iran,* In this research paper, an experimental break through curve for Salicylic acid in an adsorption recovery process was determined by a weak and strong base anion-exchange resin IRA-93 and PUROLIT A-400, respectively. Also the effect of volumetric flow rate of feed on this break through curve is studied. Generally, the results were shown that PUROLIT A-400 has larger saturation capacity of adsorption in compare with IRA-93. By increasing of feed flow rate, the amount of adsorption reduced, so that we should be determined optimize flow rate for adsorption.

14:50 ~ 16:10 (Room105)

Free Surface Flows (II)

Session Chair : Dr. C. O. Ng, Hongkong Univ/Hongkong

M-2E-1. EFFECT OF RIGID-STEM VEGETAL DRAG AND BED ROUGHNESS ON FLOOD PLAIN FLOWS

D. G. RATHNADIWAKARA, S. B. WEERAKOON, *University of Peradeniya, Sri Lanka,* Study has been carried out to investigate the roughness caused by rigid stem non submerged roughness elements representing sparse grown vegetation along with the bed roughness in the flood plain flows. A series of laboratory flume experiments were carried out using a 20m long, 0.4m wide glass walled tilting flume. Galvanized rods of 3.85mm were used to simulate the roughness elements in the channel. Different bottom roughnesses were obtained by using different surfaces of the channel bottom. Tests were conducted to find the vegetal drag coefficient under two different bottom roughnesses, by using different surfaces of the channel bottom, corresponding to the Mannings roughness coefficients of 0.012 to 0.015, and under different element patterns. Water was circulated using a recirculating pump, and discharge, depth and slopes were measured corresponding Froude Numbers ranging from 0.03 to 0.31. The results derived from more than 100 test runs are well compared with the results of the reported previous research. The flood plain flows with bottom roughness and non-submerged vegetation can be classified into three types;

- A flow region, below a certain Reynolds Number Re_{Crit1} where the flow is governed by bottom roughness. Effect of vegetal elements can be neglected. Re_{Crit1} depends on the vegetal density and bed roughness. $\mathrm{Re}_{Crit1} = \dfrac{e^{(274n+6.52)}}{\lambda^{0.577}}$, n = Manning roughness coefficient, λ is the vegetal area parameter.
- A flow region beyond a certain Reynolds Number where the flow is governed by non-submerged vegetal elements Re_{Crit2}. Vegetal drag coefficient is constant for a given pattern of non-submerged vegetal elements. Re_{Crit2} depends on the bed roughness and non-submerged vegetation density of the flood plain. $\mathrm{Re}_{Crit2} = \dfrac{e^{(121n+5.65)}}{\lambda^{0.723}}$. A flow region between Re_{Crit1} and Re_{Crit2} where the flow is governed by both vegetal drag and bed roughness.

M-2E-2. FREE-SURFACE MODELING IN A VORTEX SETTLING BASIN

A. N. ZIAEI, A. R. KESHAVARZI, H. EMDAD, *Shiraz University, Shiraz, Iran,* A three-dimensional numerical model has been developed to study the complex flow situations with air-water interface in a vortex settling basin (VSB). A code was developed to solve the Navier−Stokes (NS) equations using finite volume approach in general curvilinear coordinates. The well-known SIMPLE algorithm was implemented for velocity-pressure coupling. The free-surface motion was tracked by using volume of fluid (VOF) method based on piecewise linear interface reconstruction algorithm. The validated code was used to study the unsteady flow behavior in a circular cylindrical VSB with a central clock-wise vortex. The detailed discussions about complex three-dimensional flow patterns, velocity fields, fluid particle trajectories, and free-surface deformations in the cylindrical VSB have been presented and discussed. These help to shed more light on the very complicated flow structure in the vortex chamber.

M-2E-3. DAMPING OF CAPILLARY WAVES ON HIGHLY CURVED INTERFACES WITH EDGE CONSTRAINTS

Rangachari KIDAMBI, *CTFD Division, NAL, Bangalore, India,* We present a new eigenvalue formulation to calculate the frequency and damping rates of capillary waves on highly curved interfaces, as for example would obtain in zero-gravity conditions, in a circular cylinder with a pinned contact line. The natural viscous eigenfunctions of the geometry are used, in conjunction with the projection of the boundary conditions onto an appropriate basis, to set up a matrix eigenvalue problem which has to be solved iteratively along with a dispersion relation connecting the spatial and temporal eigenvalues. Results for high Reynolds number tend to the inviscid values previously published in literature. The behaviour of the damping rate and frequency for the lowest three non-axisymmetric modes (1,0), (2,0) and (3,0) with the contact angle θ_c and liquid depth h is explored. For a fixed depth, the frequencies show a peak around 95^0 while the damping rates decrease with increasing contact angle. For fixed θ_c, most of the variation in the frequency and damping rate occurs for the lower depths upto h = 0.9 beyond which it is almost constant.

M-2E-4. SOME RESEARCH RESULTS ON THE HYDRODYNAMIC PROBLEMS FOR FLOOD FORECASTING AND FLOOD CONTROL

N. V. DIEP, H. V. LAI, H. N. HIEN, *Institute of Mechanics, VAST, Vietnam*, N. V. HANH, *VIWRR, MARD, Vietnam,* In recent years, big floods frequently happened in Vietnam, and flood disaster causes massive losses of human life and immense damages. To reduce these damages, for short-term and long-term flood prevention and control followings measures are taken: strengthening dike systems, clearing river flows for flood discharge, building reservoirs to reduce floods in upstream of big rivers, diverting and retaining floods, reforesting and protecting watersheds, intensify dike management and protection. Before taking any of these measures, important information and data must be provided or predicted, and advanced modeling technologies are a privileged tool for such decision - making. In participating in many national and international research projects concerning the problem of integrated flood management, the Institute of Mechanics has collected and created a data base, has developed and used different modeling tools for flood control & management: database, hydrological, one and quasi two dimensional hydraulic, two dimensional hydraulic, one and two dimensional hydraulic dam & dike break flow, and socio-economic, Pilot Decision Support System coupling above indicated models. In this paper it is presented an overview on some investigations of the Institute of Mechanics in developing hydraulic models linked with the hydrologic one for flood forecasting and flood control: one- and quasi-two dimensional model for subcritical flows, one- dimensional hydraulic model for dam-break flows, two-dimensional hydraulic models and coupling between distributed overland flow and 1D & 2D hydraulic models for flash flood simulation**.**

14:50 ~ 16:10 (Room106)

Multiphase and Particle-Laden Flows (II)

Session Chair : Prof. M. Sadatomi, Kumamoto Univ/Japan

M-2F-1. NUMERICAL SIMULATION OF COLD FLOW FUEL INJECTION IN DIESEL-ENGINE CONDITIONS

M. D. EMAMI, *Isfahan University of Technology, Isfahan, Iran*, S. KHERADMAND, *Isfahan University of Technology, Isfahan, Iran,* A numerical simulation is performed for spray injection into an initially quiescent air in a constant volume chamber, using OPENFOAM software. The hybrid, Eulerian-Lagrangian formulation is used where the continuous-phase equations are written in an Eulerian form and the governing equations for the discrete, spray parcels are considered in a Lagrangian framework. A compressible version of the k-ε model is used for turbulence modeling. The simulation of spray consists of several sub-models, including the atomization model, the breakup model, and models for drag and gravitational forces. The Kelvin–Helmholtz-Rayleigh-Taylor (KHRT) breakup model is used for the simulation. Four cases are numerically simulated and compared with experimental data, as well as others' numerical predictions. The present model follows the experimental trend and could predict the spray penetration depth better than others' numerical simulation for all four cases. The differences may be attributed to the spray break-up model, as the numerical predictions at the first few time steps are close to each other, but distance from one another as time increases. At larger times, the flow induced by spray becomes more turbulent and inhomogeneous structures appear. These structures may serve as a source of error for the simulations, as we are using a very simple turbulent model that could not predict the large structures of turbulence in the background gas. The penetration length decreases as the pressure of the chamber is increased, which may be attributed to a more dense background gas and, consequently, higher drag forces acting on spray parcels at higher pressures. By comparing spray penetration at different pressures, it is seen that during the very first time span at the beginning of injection, the penetration depths of sprays are nearly the same for all