M-2C-3. OPTIMIZATION OF COMPLIANT COATING PARAMETERS FOR DRAG REDUCTION

Inwon LEE, *ASERC, Pusan National University, Korea*, Victor. M. KULIK, *Institute of Thermophysics, Russian Academy of Sciences, Russia*, Basel M. SEOUDI, *Dept. of Naval Architecture and Ocean Engineering, Pusan National University, Korea,* Ho Hwan CHUN, *ASERC, Pusan National University, Korea*, The problem of finding parameters of compliant coating providing close-to-optimal interaction with a turbulent flow becomes was advanced. To this end, the normal and longitudinal surface deformations of a flat layer of viscoelastic material glued onto a solid base were determined under the action of traveling pressure wave. Two components of the coating compliance are described by two components of the surface deformation. The dimensionless compliance depends only on the viscoelastic properties of the material, the ratio of the wavelength to the layer thickness λ/H, and the ratio of the wave velocity to the velocity of propagation of shear oscillations V/C_t. The effect of viscous losses and the Poisson's ratio on the dynamic compliance is analyzed. The maximum value of compliance is reached at $V/C_t = 1.52$ and $\lambda/H \sim 3$. For short low-velocity waves the normal compliance is several orders of magnitude larger than the longitudinal one. Owing to the increase in wave velocity and losses in the material, both components become closer to each other. These facts should be taken into account in explaining the mechanism of drag reduction determined by correlation of normal and longitudinal perturbations introduced by the compliant coating into the near-wall region. Therefore, phase relations between these perturbations are also important. Compliant coatings made of a material with Poisson's ratio less than 0.5 seem to be most effective ones, because they have greater compliance at low velocities.

M-2C-4. ASSESSMENT OF DRAG REDUCING EFFICIENCY OF OUTER-LAYER VERTICAL BLADES

Inwon LEE, *ASERC, Pusan National University, Korea*, Nam Hyun AN, *Division of Shipbuilding and Marine Engineering, Koje College, Korea,* Ho Hwan CHUN, *ASERC, Pusan National University, Korea*, The hairpin vortices in the turbulent boundary layer are plays a main role in the self-regeneration mechanism of the coherent structures and is considered to be a major source of the turbulent skin-friction. Various turbulent flow control strategies are based on the disturbance as well as the attenuation of those regeneration mechanisms of the coherent structures. Recently, up to 30% of the turbulent skin-friction reduction by installing an array of vertical blades, which were subsequently named as outer-layer vertical blades, has been reported. The drag-reducing efficiency of the outer-layer blades, however, was quantified only in terms of the reduction in the local skin-friction coefficient, which was measured from the local velocity distribution. The assessment of the drag reducing efficiency is mainly restricted to the downstream region of the blades. Indeed, sufficient care has not been taken to such adverse effects as the increase in the wetted surface area and the flow disturbances due to the presence of the blades. In other words, verification of the drag reducing efficiency in terms of the total drag force remains to be unsolved yet. In the present study, a series of drag force measurements has been performed toward assessments of the total drag reducing efficiency of the outer-layer vertical blades in towing tank and circulating water channel. The major design variables such as chord length, height and span-wise spacing were set to equal to those values in the previous research in normalized size by wall variables.

14:50 ~ 16:10 (Room104)

Industrial Applications and Material Processing Flows (II)

Session Chair : Prof. S. J. Lee, POSTECH/Korea

M-2D-1. ANALYSIS OF TRAIN-INDUCED FLOW THROUGH SUBWAY VENT SHAFT RELATING WITH SHAPE CHANGE OF VENT SHAFT

J. Y. KIM, *Korea Institute of Construction Technology, Korea,* As traffic increases drastically due to industrialization and urbanization, the subway, with superior punctuality and speed, has become the main transportation system in Korea as well as in other major countries. The subway is considered as a life space of citizens beyond simple concept of transportation, and the request for safety and comfort is gradually increasing. But, shaping as semi-closed tunnel, the subway is vulnerable to fire disaster, poor IAQ and thermal discomfort. So, it is strongly required to develop the technology for the improvement of subway safety and environment. For above reason, the ventilation and smoke-control systems are designed and the vent shaft and air blower are installed in subway tunnel. These ventilation and smoke-control systems could be improved for better performance through aerodynamical study. In the mean time, in process of city's growth, development and extension, it would happen that new roads are designed upon the vent shaft of operating subway. To accept the road construction, the vent shaft should be changed to be outside new roadway and the geometry of air flow passage in vent shaft also will be transformed to other shape. To conserve the environment of subway tunnel, it is necessary to present the shape-changing method which makes the air flow rate through the vent shape not to decrease. In this study, three-dimensional numerical analyses of natural ventilation system in operating subway are carried out relating with the different air flow passages of vent shaft which are before and after the construction of new road. And the train-induced air flow rate through the vent shaft is measured in operating subway to assess the ventilation performance of shape-changing vent shaft. The following conclusion were drawn. (1)The flow rate of outgoing air through the vent rises as the train approachs the vent shaft, and when the train passes the vent shaft, the flow rate starts to decrease. After the train completely passes the vent shaft, the outdoor air comes into tunnel and the flow rate increases again. (2) For the vent that remain unchanged, the flow rate through the vent at the time of after-construction is more than at the time of before-construction. For the vent that has longer flow passage length after the shape-change construction, the flow rate through the vent at the time of after-construction is less than before-construction. (3) After the vent shaft undergoes the shape-change construction, the overall flow rate through vent shaft owing to the train-induced air flow decreases by about 4.2%.

M-2D-2. NUMERICAL INVESTIGATION OF THE FLOW AND HEAT TRANSFER CHARACTERISTICS IN A CHANNEL WITH PERIODIC TRANSVERSE GROOVES

W. CHANGCHAROEN, *MUT, Thailand,* S. EIAMSA-ARD, *MUT, Thailand,* P. PROMVONGE, *KMITL, Thailand,* This work presents a numerical investigation of flow field and heat transfer characteristics in a rectangular channel with grooves on one broad heated wall. Governing equations for the steady turbulent 2D flow are solved numerically under certain boundary conditions. The numerical computation encompassed the Reynolds number in a range of 12,000 to 22,000 for five different shapes of periodically grooved parts (c/e = 0.0, 0.5, 1.0, 1.5, and 2.0). The effect of important parameters on the heat transfer coefficient and friction factor has been discussed and the results are compared with that of smooth wall channel under similar flow conditions. The Renormalized Group (RNG) $k-\varepsilon$ turbulence model is selected after comparing its predictions with the shear stress transport (SST) turbulence model due to better agreement with experimental results available in the literature. The present investigation clearly demonstrates that the heat transfer and friction factor for grooved wall is higher than that for smooth wall, especially at c/e = 1.0. The conditions for best thermal performance have been determined.

M-2D-3. AN INVESTIGATION OF THE THERMAL HEAD EFFECT IN A POWER TRANSFORMER

C. H. CHA, W. S. KIM, K. Y. KWEON, *Power & Industrial Systems R&D Center, Hyosung Corporation, Korea*, C. H. KIM, G. J. YOO, H. K. CHOI, *Chang won National University, Korea*, This paper describes the numerical simulation for improving heat transfer in a power transformer by the difference in elevation between the center of the coils and the center of the radiators which was called the thermal head. The objectives of this study are to investigate the thermal head effect for improving the heat transfer characteristics of the radiator in a power transformer using the Fluent code and to have a clear understanding of the cooling pattern in a radiator which is a main heat remover in the power transformer for optimizing the radiator design increasing the thermal efficiency. We analyzed the flow rate variation due to the increase of thermal head in the ONAN, ONAF cooling mode. The simulations were performed to analyze the thermal head effect in a power transformer using commercial CFD code (Fluent 6.3) and to optimize the design of radiator for improving heat transfer. For the ONAN cooling mode, if there is 90% thermal head increase which mean 38% heat transfer area reduced in a radiator, the flow rate in experiments was increased about 40% and 70% for calculations. For the ONAF, it can be seen that the increased flow rates for experiments and calculations are lower than the reduced heat transfer area in comparison with the ONAN cases. Although the effect of thermal head is still to increase flow rate, in practice, the cooling fan effect dominates with the result that the effectiveness of flow rate is improved. It is clear that in the design of power transformer the thermal head would significantly help to minimize the cost and size of radiators.

M-2D-4. MASS TRANSFER AND DYNAMYC INVESTIGATIONS OF SALICYLIC ACID ADSORPTION ON TO ANION-EXCHANGE

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. $\mathrm{Re}_{Crit1} = \frac{e^{(274n+6.52)}}{\lambda^{0.577}}$, 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. $\mathrm{Re}_{Crit2} = \frac{e^{(121n+5.65)}}{\lambda^{0.723}}$. 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