substrate where individual molten particles impact, cool and solidify to form a deposit. This technology is used to produce coatings for wear, thermal, oxidation, and corrosion protection. In this paper, a 3-D stochastic model is used to simulate the coating morphology in a thermal spray coating process. Four main assumptions used in the stochastic model are: the spray droplets are non-interacting point particles; each droplet has a different size, velocity, and impact position; the spray is random; and the probability of obtaining a droplet occurrence at any instant is independent of other droplets occurring at other instants. It is assumed that the position of droplet impact follows the uniform distribution and the droplet specified diameter and velocity follow the Poisson distribution. The splashing and rebounding of droplets during the impact are not considered in this study. A set of rules are used to specify the final splat shape as a function of droplet impact conditions. These rules obtained from the literature are based on the numerical/analytical solution of the droplet spreading and solidification. Final splat shapes are characterized by dimensionless numbers known as Reynolds, Weber and Stefan. Due to temperature difference between droplet and substrate and thermal stresses after solidification, the edge of the splats is curled up. A new analytical model is used for this curl-up mechanism. The curl-up is assumed to be the sole reason for porosity formation. Simulations were performed for a small section of a substrate on which alumina droplets are sprayed. The computed thickness and porosity were in good agreement with those reported in the literature. In another simulation for aluminum droplets impinging on a steel substrate, the results for porosity were found in the range measured in experiments. The effect of substrate temperature on the porosity was also investigated. The results from both experiments and model show that by increasing the substrate temperature, the porosity increases. There were some discrepancies between the two results, however, that could be attributed to the existence of droplet splashing ignored in the model. The effect of spray materials on the coating porosity was also studied. The coating formed from the spray of alumina particles on a steel substrate had the lowest porosity and that of the nickel particle had the highest. The difference in porosity values for various materials can be attributed to droplet physical properties namely the surface tension and viscosity.

T-2F-2. A NUMERICAL MODEL FOR CALCULATION OF THE FORM AND VELOCITY OF LONG BUBBLES IN TUBE AT NEGLIGIBLE GRAVITY USING BOUNDARY ELEMENTS METHOD

HIEN Ha-Ngoc, *Institute of Mechanics, Hanoi, Vietnam,* The paper presents a numerical model for calculation the form and the velocity of long bubbles under micro-gravity conditions or in sub-millimeter tubes when surface tension dominates. To understand the role of surface tension, the numerical method is developed in frame of the inviscid theory. The bubble is assumed to be axi-symmetric and move at constant velocity with a prescribed velocity profile of liquid ahead the bubble. Then, the flow characteristics can be described by a Poisson equation for the Stockes stream function. An equation resulting from both Bernoulli equation and the pressure jump conditions at the interface is obtained and used for determining the bubble shape. The boundary elements method (BEM) was used to solve the problem in an iterative way to obtain simultaneously the flow characteristics, the bubble velocity and shape. The obtained results by the model are in good accordance with experimental results for the limit case of large bubble Reynolds number.

T-2F-3. FLOW INSIDE A DROPLET MOVING ON A FLAT SURFACE

A. HAYASHI, *Toyo University, Japan*, O. MOCHIZUKI, *Toyo University, Japan,* The purpose of this study is to investigate an entrainment mechanism of particles of dust into a droplet moving on a flat surface. This is useful for developing a way to clean a surface by using a droplet. The entrainment of dust into a droplet may be affected by conditions of a flow and surface of a wall. The flow inside a water droplet moving on a flat surface was visualized by starch particle. The droplet ran from left to right on the acrylic resin surface inclined 20 degrees. The speed of the droplet was 0.01 m/s. Its volume was 0.1×10^{-6} m^3. Its size was the length in the moving direction 10×10^{-3} m, width 5.4×10^{-3} m and height 2.5×10^{-3} m. The Reynolds number when assumed the width of the droplet representative dimensions was about 50. The dominant motion of particles observed in the side view picture was clockwise rotating flow. Particles near the wall were getting together, and were moving toward the rear along the center of the droplet. The flow patterns were topologically considered to know three-dimensional structures of the flow. There is a half vortex ring in the moving droplet. The half nodes are presented at both the front and rear positions on the wall in a droplet. The entrainment of starch particles dispersed on the surface of the wall. The droplet ran on the particles. It was found that the particles were captured only at the rear of the droplet. This is our important result.

T-2F-4. NUMERICAL SIMULATION OF FLOW INSTABILITIES DURING THE RISE OF A BUBBLE IN A VISCOUS LIQUID

Mohammad P. FARD, Mehran M. FARHANGI and Hossein MOIN, *Department of Mechanical Engineering, Ferdowsi University of Mashhad, Mashhad, Iran,* In this paper, the flow instabilities during the rise of a single bubble in a narrow vertical tube are studied using a transient 2D/axisymmetric model. These instabilities include the oscillation of the bubble shape and formation of a wake behind it. In the model, the Navier-Stokes equations in addition to an advection equation for liquid volume fraction are solved. A modified Volume-of-Fluid (VOF) technique based on Youngs' algorithm is used to track the liquid/gas interface. As a first step the model was subjected to several tests in order to validate its results. The results of simulations for terminal rise velocity and bubble shape are compared with those of the experiments. The results of the model, are predicted in the same region where observed by experiments. The results show that increasing the bubble diameter increases the rise velocity up to a certain limit after which the bubble starts to oscillate. In this regime, the rise velocity remains nearly constant. Further increase of the bubble diameter changes the deformation behavior to the spherical cap regime. Next we studied the flow instabilities that occur during the rise of a bubble in a narrow vertical tube. Driven by the buoyancy force, the bubble rises rapidly after its release. It is deformed from the initial spherical shape to the final bullet-like configuration. The bottom of the bubble moves rapidly upward and develops into a concave shape. It then rebounds downward immediately into a convex shape. This up-and-down oscillatory movement of the bubble bottom continues as the bubble rises with decreasing amplitude. The top of the bubble, on the contrary, remains a spherical cap shape with very little deformation as it ascends. Finally the effect of different parameters on the oscillatory behaviors of bubble velocity and shape are investigated.

14:30 ~ 15:50 (Room 107-108)

Computational Fluid Dynamics (V)

Session Chair : Prof. Y.-W. Lee, Pukyong Univ/Korea

T-2G-1. AN OIL SPILL MODEL FOR NORTHERN PERSIAN GULF WATERS

M. A. BADRI, A. R. AZIMIAN, *Department of Mechanical Engineering, Isfahan University of Technology, Iran,* In this paper, simulation of oil spill due to weathering and tidal currents in Persian Gulf is studied. Here, water current and wind-induced velocities are taken into account including many significant processes such as advection, surface spreading, evaporation, emulsification and dissolution. A grid with 339 points in the Persian Gulf have been generated. By means of WAve Model (WAM) and Cressman analysis on the whole grid, wind velocity and direction, wave height and wave period have been determined. Tidal constituents have been obtained from co-tidal charts and then tidal stream from tidal analysis program have been calculated to determine advection properties. Therefore, a portal have been provided to present simulation of the surface movement of the oil slick by Lagrangian approach for the northern part of the Persian Gulf waters. Sample simulations for oil spill are presented and the results are compared with the existing observed data. Comparison of wind and tide data and water surface level with the observed data and some other simulation results shows good agreement.

T-2G-2. NUMERICAL STUDY OF THE GAS FLOW THROUGH A CRICAL NOZZLE

S. MATSUO, *Saga University, Japan*, T. MITSUNAGA, *Saga University, Japan*, T. SETOGUCHI, *Saga University, Japan*, H. D. KIM, *Andong National University, Korea,* The critical nozzle is defined as a device to measure the mass flow with only the nozzle supply conditions, making use of the flow-choking phenomenon at the nozzle throat. The mass flowrate and critical pressure ratio are associated with the working gas consumption and the establishment of safe operation conditions of the critical nozzle. According to previous researches, the mass flowrate and critical pressure ratio are strong functions of Reynolds number. Some studies have shown that for high Reynolds numbers, based upon the velocity at the nozzle throat and the diameter of the nozzle throat, the discharge coefficient approaches unity, indicating that the one-dimensional theory is valid for the prediction of the mass flowrate. However, for lower Reynolds numbers, it reduces to considerably below unity, attributing to the wall boundary layer effects on the mass flowrate through the critical nozzle. In the present study, the effects of amplitude and frequency of back-pressure fluctuations in the downstream of the critical nozzle exit on the flow field close to the nozzle throat were clarified numerically. The results obtained from the steady computations were in close agreement with the previous experimental data. In order to simulate the effects of back-pressure fluctuations on the critical nozzle flow, an excited pressure oscillation was assumed downstream of the exit of the critical nozzle. The computed results showed that, for Reynolds numbers below Re = 3740, the unsteady effect of the pressure fluctuations can propagate upstream of the nozzle throat and it has effects on fluctuations in mass flow rate through the critical nozzle, even in chocked-flow conditions. In cases of the pressure disturbance with large amplitude for Re = 7470, the pressure fluctuations can propagate upstream of the nozzle throat. However, it is found that the mass flowrate does not fluctuate with time.

T-2G-3. NUMERICAL INSTABILITY ANALYSIS OF THE LATTICE BOLTZMANN EQUATION METHODS USING DIFFERENT SCHEMES

A. R. RAHMATI, *Isfahan University of Technology, Iran*, M. ASHRAFIZAADEH, *Isfahan University of Technology, Iran*, E. SHIRANI, *Isfahan University of Technology, Iran,* The lattice Boltzmann equation (LBE) method has been recently developed into an alternative effective tool to simulate fluid flows. Although it has significant advantages over the conventional CFD methods, there are still some restrictions in the utilization of LBE models. One of the shortcomings is that the physical boundary conditions for macroscopic variables cannot be implemented directly since the dependent variable in the LBE model is the density distribution function. Furthermore, for the Single-Relaxation–Time Lattice Boltzmann Equation (SRT-LBE) model when the Reynolds number is large, the relaxation parameter approaches to the stability margin if the number of mesh points is not large enough. The instability problems may be compounded in three-dimensional flows when physics may not be adequately resolved owing to computational constraints. Much progress has been made in this direction in recent years and several approaches have been proposed to increase numerical stability. Some of such approaches are Entropic Lattice Boltzmann Equation (ELBE) method, Multi-Relaxation-Time Lattice Boltzmann Equation (MRT-LBE) method and Fractional Volumetric Lattice Boltzmann Equation (FV-LBE) method. In the present work, numerical stability of the ELBE, MRT-LBE, FV-LBE models and a combination of the two last methods, i.e., FV-MRT-LBE model are compared to that of the SRT-LBE method for the simulation of the lid-driven cavity flow at different values of the Reynolds number, ranging from 1000 to 10000, on a 257× 257 grid. The FV-MRT-LBE model is a new approach that is presented in this work and its results are compared with the results obtained by other methods. Results show that the stability and accuracy of all methods are comparable with that of the SRT-LBE method at lower Reynolds numbers. However, as the Reynolds number is increased, the stability of all methods turns out to be better than that of the SRT-LBE method. Furthermore, the MRT-LBE method produces more stable and slightly more accurate results compared to ELBE, FV-LBE, and FV-MRT-LBE methods, especially at higher Reynolds numbers.

T-2G-4. ABILITY OF MIXING TWO IMMISCIBLE LIQUIDS IN A KENICS STATIC MIXER

J. Y. C. LEONG, *Monash University, Malaysia*, C. F. THAN, *Monash University, Malaysia*, Y. W. OOI, *Monash University, Malaysia,* The ability of the Kenics static mixer, a commercially available mixer that does not involve the use of mechanical agitation, in mixing of two immiscible liquids has been investigated. The model object is a length of pipe containing the Kenics static mixer elements each resembling a short helix. The two modeled phases were palm oil triglyceride in the continuous phase and methanol in the discrete phase. This process mimics the pre-mixing process before the transesterification process which takes place in the biodiesel production. The flow field was simulated numerically using the commercially available Computational Fluid Dynamics software package, Fluent. In the preprocessing phase, the modeling of the model object incorporated both tetrahedral and hexahedral meshing schemes to discretize the model geometry. Solving the flow fields involved choosing the appropriate models. These included choosing between the various laminar and turbulent models, an appropriate multiphase model as well as an appropriate drop breakage model. Postprocessing was concerned with extracting relevant information from the flow field to evaluate the performance of the Kenics static mixer and involved assessing its effectiveness in mixing the two fluids by statistical means and estimating the pressure drop across the mixer. Simulated data predicted that the Kenics mixer with an RL (successive elements were installed with an alternate twist) configuration is the most effective static mixer. On the other hand, the Kenics RR (successive elements were installed in the same direction) mixer is more efficient as it achieves mixing with less power requirement. A better understanding of the Kenics static mixer will enable its incorporation into the production cycles that necessitates the dispersion of two immiscible liquids to affect the kinetics of reactions as seen in the transesterification process in biodiesel production.

16:00 ~ 17:20 (Room 101)

Flows with Heat Transfer (I)

Session Chair : Prof. J. S. Park, Halla Univ/Korea

T-3A-1. HEAT TRANSFER ENHANCEMENT OF RECTANGULAR RIBS WITH CONSTANT HEAT FLUX LOCATED IN THE FLOOR OF A 3D TURBULENT DUCT FLOW

E. ESMAEILZADEH, A. ALAMGHOLILOU, H. MIRZAIE, A. KHOSHNEVIS, *Department of Mechanical Engineering, the University of Tabriz, Tabriz, 51666-14766, Iran,* In this paper numerical investigation of hydrodynamic and forced convection heat transfer in a rectangular horizontal duct has been undertaken. Heat sources are cross rectangular ribs with small aspect ratio and uniform heat flux under turbulent regime. The purpose of this study is to apply a passive method for increasing rate of heat transfer from the ribs. Geometry and the physic of the problem are roughly similar to cooling of electronic boards. Therefore three rectangular ribs established along the width of the channel with specified distance on the floor. Between ribs some vortexes are appearing which in general act like a heat traps and reduce the heat transfer rate. This thermal resistance should be neutralized by applying heat transfer enhancement methods. Establishing holes between the ribs because of that the interior pressure is much less than environment is an advantageous method which causes distortion of vortexes and finally augmentations of heat transfer by producing a secondary flow without any outsource energy. This method classifies as passive methods. Numerical simulation for assumed geometry is performed by solving governing equation in finite volume with Phoenics software. Obtained results were compared with available experimental results of literature and indicate good agreement. Comparison between plain case and passive case shows effectiveness (PEC) is related to geometry parameters significantly specially to number of holes and their arrangements. In this investigation 9 arrangements was analyzed so results compared and discussed completely.

T-3A-2. STUDY OF THE EFFECT OF PARTICLE SIZE ON THE HEAT TRANSFER IN A FLUIDIZED BED HEAT EXCHANGER

Y. D. JUN, *Kongju National University, Korea*, K. B. LEE, *Kongju National University, Korea*, S. Z. ISLAM, *Kongju National University, Korea*, S. B. KO, *Kongju National University, Korea*, and M. F. KADER, *Kongju National University, Korea,* Heat recovery from flue gas from industrial furnaces, boilers and incinerators for better use of energy resources is a nation-wide concern in Korea. To overcome the fouling of fly ash on the heat transfer surface and erosion and periodical cleaning which are the major drawbacks in conventional heat exchangers for flue gas heat recovery, a single riser no-distributor-fluidized-bed (NDFB) heat exchanger is devised. Compared to the existing ones, the present heat exchanger system is featured in the particle fluidization method which does not depend on conventionally used baffle plate with holes and the multiple down comer tubes for heat extraction from the heated particles. The heat transfer performance and pressure drop, effect of suspension density and particle size is studied for this no-distributor-fluidized-bed (NDFB) heat exchanger system. It was observed that the effect of particle size on the heat transfer is more significant for smaller particles and larger suspension densities.

T-3A-3. RE-CIRCULATION BEHAVIOUR IN THE FLOW FIELD OF A LOW ASPECT RATIO DUMP COMBUSTOR

N. P. YADAV, *IIT Kanpur, India*, A. KUSHARI, *IIT Kanpur, India,* This paper reports an experimental investigation of the recirculation behaviour inside a low aspect ratio dump combustor. The length of the combustor studied was less than the reattachment length for the separated flow. The exit of the combustor is tapered which supports the flow reversal from the exit section. The recirculation behaviour in the combustor is evaluated from the cold flow visualization and pressure measurement studies. This recirculation was induced by the geometry of the combustor, therefore, it was an unforced recirculation. In this study, a dissimilar thickening of the recirculating flow was observed inside the combustor that happened due to the flow reversal from the exit section. The recirculation and flow reversal from the tapered exit section encourage a cyclic behaviour in the vortex. The findings of the visual study were corroborated by measuring the wall pressure distribution and the results were found to be in a good qualitative agreement with each other. The findings of this study can have applications