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