LARGE EDDY SIMULATION OF WAVE BREAKING WITH MOMENTUM-ADVECTED SCHEME FOR UNI-PHASE BUBBLY FLOW

Yuriko Matsubayashi [1] and Akio Okayasu [2]

In order to simulate wave breaking process, air mass and bubbles entrained in water should be considered in calculation. In the present study, a numerical scheme is developed for wave breaking assuming an incompressible uni-phase flow with density varying between air and water. To satisfy the momentum conservation over the computational domain with large density variation, Navier-Stokes equation described in terms of momentum advection is employed as the governing equation. The advection terms of the momentum equation and advection equation for Density Value Method are solved by R-CIP method to minimize numerical diffusion. The results are compared with experimental results for a dam-break, bore and wave breaking on a slope. It is found that the model can reproduce bubbly areas due to wave breaking reasonably well.

Keywords: wave breaking; Large Eddy Simulation; bubbly flow; momentum conservation

INTRODUCTION

Wave breaking associated with rapid change of surface boundary and air entrainment is one of the most complicated phenomena in air-water interface problems. Computational Fluid Dynamics (CFD) is now frequently employed to reproduce wave deformation and flow fields under wave breaking. For example, Miyata et al. (1996) did simulation of three-dimensional breaking waves with a modified marker-and-cell (MAC) method. Watanabe and Sasaki (1999) tried to describe fluid motion of turbulent flow and eddy generation by wave breaking using large eddy simulation (LES). Kawasaki (1999) calculated wave breaking over a submerged breakwater with volume of fluid (VOF) method. As wave breaking is motion of turbulent flow with air entrainment and bubble generation, air-water two-phase models are often used to simulate flow associated with wave breaking. Mutsuda and Yasuda (2000) confirmed that a three-dimensional two-phase flow model with C-CUP method could reproduce air entrainment for wave breaking simulation. With applying multi-phase flow models, it is expected that complex flow motion with large-scale vortex, air entrainment, bubble and splash generated by wave breaking can be reproduced. Improvement of computer technology may make it possible to evaluate complex nearshore phenomena, such as coastal morphological change and material transfer between atmosphere and ocean, by numerical simulations.

As for the study of fluid motion with bubble entrainment, Ma et al. (2012) calculated single bubble plume with the VOF method and showed that their model of the Eulerian method could calculate rising bubble behavior with using sufficiently fine grids which were smaller than 0.25mm for 4mm diameter bubbles. Lubin et al. (2006) carried out gas-liquid two-phase simulation with treating boundaries between gas and liquid with a CSF model and found that for rising bubble calculation with grids of one tenth of a bubble size was at least needed. However, the bubbles generated by wave breaking have wide variation in their sizes from large entrained air to micro bubbles. Mori and Kakuno (2008) found for their laboratory experiment that the averaged diameter of air bubbles in breaking waves was around 0.5cm. Considering the result of Lubin et al (2006), grid size required for calculation of moderate bubbles under breaking waves in laboratory is less than 0.5mm, which is far smaller than the representative scale of wave motion. It means that it is not realistic to solve motion of complex wave fields with mesh sizes enough fine to simulate bubble behavior as described above.

About simulations of bubbly flow, some used models in which effect of bubbles smaller than the grid size was considered in macroscopic manners. Watanabe et al. (2009) modeled air bubbles smaller than the grid size in their SGS model and calculated interaction between behavior of bubbles and ambient flow. Gong et al. (2007) carried out a numerical simulation for ozone dissolution in a bubble plume and estimated optimal bubble size for dissolution. However, in wave breaking zones, bubble size shows wide distribution from bulky air entrained at initial stage of wave breaking to micro air bubbles, and breakup and coalescence of bubbles keep changing the bubble size distributions. Thus it is inevitable for a bubble to change its diameter across the size of calculation grids. A new simulation

---

[1] Department of Civil Engineering, Iwate University, 4-3-5, Ueda. Morioka-City, Iwate, Japan.
[2] Department of Ocean Science,Tokyo University of Marine Science and Technology, 4-5-7, Konan , Minato-Ku, Tokyo, Japan.

method simultaneously covering air mass larger than the grid resolution and sub-grid size micro air bubbles should be developed to simulate the complex behavior of bubble with wide size- variation.

In the context described above, a rather simple macro model which represents bubbly flow as a mixture of air and water is presented in the present study. The model is described in a frame of Eulerian method and it does not explicitly give the air-water boundaries. The flow is calculated as a uni-phase flow with its density changes 1,000 times. The model may not be a sophisticated way of calculation at least for micro air bubbles, but it can be a feasible method of wave calculation considering air entrainment by wave breaking.

With Eulerian methods the momentum conservation and the mass conservation are to be satisfied by applying the Navier-Stokes equation and the continuity equation at every grid space fixed on the coordinates. A simple discretization of the Navier-Stokes equation, however, gives certain amount of error in momentum conservation for flow with density variation, because in general density, velocity and those gradients are not given at the identical points in the calculation. Inconsistencies among distributions of those variables results in momentum errors in calculation with discretization based on the ordinary Navier-stokes Equation. Thus with the most of Eulerian methods, the momentum is not conserved for flow with large density variation and error control is difficult. In the present study, a derivative expression of the Navier-Stokes Equation which is described in terms of momentum advection is employed under assumption of incompressible fluid to minimize the momentum error. A water column collapsing, a dam-break, a bore wave and a solitary wave breaking on a slope are simulated by the method and compared with experimental results.

MODEL DESCRIPTION

Governing Equations

In the present study, flow of water, air and bubbly flow which has density between those of water and air is calculated. Generally in multi-phase flow simulation, air is treated as compressible fluid. However, for wave breaking near free surface, fluid compressibility is not a major factor for simulating flow field except cases calculating impact forces on structures. Thus in the current study, air, water and bubbly flow are all considered to be incompressible. For incompressible fluid, the volume conservation equation can be applied as one of the governing equations.

For calculation of incompressible flow, Eqs. (1) and (2) are frequently used as governing equations.

$$\frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} = -\frac{1}{\rho} \nabla p + \nu \nabla^2 \mathbf{u} + \mathbf{g} \qquad (1)$$

$$\nabla \cdot \mathbf{u} = 0 \qquad (2)$$

where $\mathbf{u}$, ρ, p, ν and $\mathbf{g}$ are velocity vector, density, pressure, kinematic viscosity coefficient and gravity acceleration vector. Eq. (1) is the Navier-Stokes equation and Eq. (2) is the continuity (volume conservation) equation. These governing equations are solved numerically with satisfying the Poisson equation for pressure, Eq. (3), where superscript indicate time level. Time level $n+1$ means next time step and F means intermediate velocity value upgraded with the second and third terms in the left side of Eq. (1).

$$\nabla^2 p^{n+1} = \frac{\nabla \cdot \mathbf{u}^F}{\Delta t} \qquad (3)$$

In order to calculate free water surface which gives large gradient (actually discontinuous change) of density in the uni-phase flow simulation, a compact grid system should be employed. In the present model, the staggered-grid system in which calculation points of pressure and density are set on center of grids and those for velocity and momentum are set on sides of grids is used. If Eq. (1) is discretized with the staggered-grid system, significant error for momentum conservation may be observed for flow with large density change because of inconsistencies among calculation points of density and velocity. To avoid this disadvantage, Eq. (4) instead of Eq. (1) is applied to the model. Since Eq. (4) is explicitly described in terms of momentum, simulation results are expected to show better conservation for momentum. Eq. (5) obtained from Eq. (4) by substituting Eq. (2) into the Eq. (4) is discretized for calculation.

$$\frac{\partial \rho \mathbf{u}}{\partial t} + \nabla \cdot (\rho \mathbf{u}\mathbf{u}) = -\nabla p + \mu \nabla^2 \mathbf{u} + \rho \mathbf{g} \qquad (4)$$

$$\frac{\partial \rho \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla (\rho \mathbf{u}) = -\nabla p + \mu \nabla^2 \mathbf{u} + \rho \mathbf{g} \qquad (5)$$

Effect of turbulence generated mainly by wave breaking is considered by applying a turbulence model to evaluate eddy viscosity in Eq. (5). In the present study, Large Eddy Simulation (LES) with the Smagorinsky model for the Sub Grid Scale (SGS) model is used.
.

Advection and Discretization

In the simulation, momentum at each grid is obtained as a result of time-evolution calculation of the momentum advection term in Eq. (5). With employing the staggered grid system, momentum and velocity for one direction are calculated at the same point on one side of the grid, but density is defined at a half grid different position (the center of grid) from the point. To obtain velocity at each time step, momentum is divided by density. Since small error of calculated velocity soon leads instability of calculation, density values implicitly included in momentum values at sides of grids should be accurately estimated from those defined at grid centers. It is however difficult to evaluate density values at a half grid away from defined positions for fields with large density variation. For example, simple linear interpolation (giving 0.5) of two density values at positions having water surface between, one in air and the other in water, does not give an accurate (or real) value (0 for air or 1 for water) of density for a point at the middle of them.

In order to mitigate this problem, CIP (cubic-interpolated pseudo-particle) method, one of methods calculating advection terms accurately, is employed for the advection calculation of the present study. To calculate advection terms with CIP method, a value at the point to be updated, a value at the next point in the upwind direction and a gradient at the upwind point are used. One advantage of CIP method is that these three values are all positioned within a length of a grid; that means "compact". This feature of CIP method makes it possible to calculate advection at regions with drastic density change, such as a field with a boundary of water and air. Another advantage is that CIP method assumes variation of the target valuable between the updated point and the upwind point as a cubic function. A value at any position in the grid can be directly and consistently obtained from the function.

A disadvantage of CIP method is that advection calculation with it easily generates overshoots and undershoots. At a calculation area with drastic density change, the overshoots and undershoots cause the error for density fields and instability of calculation. To control these errors and avoid instability of calculation, R-CIP method (rational CIP) is installed in the calculation. Since overall accuracy of advection calculation with R-CIP method at an area with drastic density change may become worse than that with the basic CIP method, Type-C multi-dimensionalization (Yabe et. al., 2004) is applied to control numerical diffusion.

Bubbly Flow Model and Air-Water Separation

On the surface of a breaking wave, bubbly flow regions are created and it is difficult to make then apart for air and water, especially in cases that the calculation grid is larger than the bubble diameter. For example, a breaking wave with a surface roller shows a bubbly flow area which spreads widely in front of the wave and it is even difficult to define the water surface either by visual observation or by measurement with a wave gage. The situation is common for problems related to overtopping on seawalls or swash on beach.

In the most of Eulerian simulation methods, position of water surface is updated by using velocity calculated with the governing equations, the Navier-Stokes equation and the equation of continuity. The position of free surface is frequently assumed in the target grid as a plane or a curved surface. In this context, water surface boundary defined with an Eulerian description method cannot describe water surface fluctuation, such as a bubble or a splash, whose scale is smaller than the grid size. However, the entrained bubbles change average density of fluid, drive ascending current or eddy motion. The interaction between the bubbles and surrounding fluid play important roles for complicated flow generation by wave breaking.

In this study, void ratio is used to describe total amount of air in a grid. Density function value F which is equivalent to $(1 - \text{void ratio})$ is introduced after Watanabe and Saeki (1999) to calculate void ratio. Density function value varies from 0 (air only) to 1 (water only) and is defined at center of each calculation grid. A value between 0 and 1 shows mixture of air and water, the ratios of which are $1 - F$ and F, respectively. The compressibility is not considered in this study, averaged density in a grid is expressed as,

$$\rho = F \cdot \rho_{water} + (F - 1.0)\rho_{air} \qquad (6)$$

where ρ_{water} is density of water, and ρ_{air} is density of air.

As for the viscosity, inter mediate value between water and air for a grid is calculated according to the ratio given by density function value with Eq. (7) for simplicity.

$$\mu = F \cdot \mu_{water} + (F - 1.0)\mu_{air} \qquad (7)$$

where μ_{water} and μ_{air} are viscosity of water and air, respectively.

Time evolution of density function value is calculated with solving an advection equation, Eq. (8) which actually works as the mass conservation equation in the present model.

$$\frac{\partial F}{\partial t} + \mathbf{u} \cdot \nabla F = 0 \qquad (8)$$

For actual bubbly flow, bubbles in fluid rise due to buoyancy and water droplets fall by gravity. This process results in separation of air and water over time. To describe bubbly flow separation in our simulation, a simple separation model is employed and change of density function value associated with the separation is calculated. By the separation model, a part of density function value F at a grid is moved to the next grid just below the grid with satisfying mass conservation. When F value is moved, the momentum values are re-calculated at both grids to satisfy the momentum conservation. The rate of separation is considered as same as the free fall of water droplets.

In this study, accurate momentum conservation around free surface is considered by calculating water, air and bubbly flow (intermediate density area) continuously. However, effects of bubbles such as generation of upwelling velocity or production of turbulence are not considered and left for future studies.

CALCULATION RESULTS

Water Column Collapsing

In order to compare simulation results calculated by the ordinary velocity-advection method in which Eq. (1) was discretized and the momentum-advection method used in the present study, a water column collapsing simulation was performed. In the simulation the density function value which took continuous value between 0 and 1 was calculated.

Figure 1 shows results of two-dimensional simulations. Height and width of calculation domain are 0.5 m and 0.5 m and the grid size is 0.01 m by 0.01 m which may be relatively large for this kind of simulation. A water column which is 0.1 m in width and 0.2 m in height is initially set at the left-bottom corner of domain. Density of water is 1000 kg/m³ and that of air is set to be 1 kg/m³ for simplicity. As shown in Fig. 1, the two simulations show almost identical results. However for the result of velocity-advection case, at right-upper corner of the original water column, lag of water fall (a small amount of residual mass of water) is observed. With the velocity-advection scheme, velocity value in air next to an air-water boundary (As it is not given by a clear surface but expressed as a large change of density value) affects water area directly through velocity-advection effect, even though there is large density difference between air and water areas. On the other hand, with the momentum-advection scheme, influence by large air velocity to the water area is quite minor because advected momentum for air is only 0.001 of that of water.
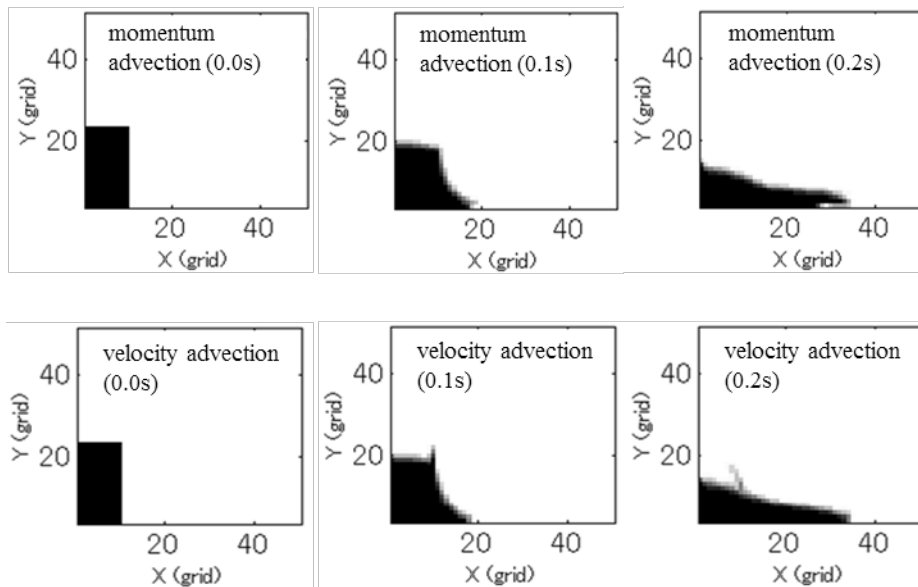
Figure 1.   Comparison between momentum-advection and velocity-advection schemes of calculation for water column collapsing

Dam Break

Simulation results with momentum advection are verified through experimental results of Janosi et al. (2004) and experiments we carried out. The experimental result by Janosi et al. (2004) has been already used for verification of many fluid simulations.

Calculation condition is shown in Fig. 2. A gate separates a section of 0.38 m long and 0.15 m deep at an upstream area from a 1.0 m long and 0.018 m deep section in a flume. When the gate is opened, water column falls with breaking to the downstream area. In our simulation, thickness of the gate is assumed to be zero, and gate opening speed is set to be 1.5 m/s. The grid size is 0.002 m by 0.002 m. Figure 3 shows a comparison of the simulation result (left panels) with the experimental result carried out by Janosi et al. (right panels). Distribution of density function value is shown for the simulation results. If F of a grid is 1.0, water is in the grid and the grid is colored by black. If F is 0.0, air is in the grid and colored by white.

The collapse of water column forms plunging-type wave breaking ($t = 0.28$ s to 0.34 s) and a small secondary break is generated by its plunging jet ($t = 0.41$ s). The secondary breaking and a mass of air entrained by the first wave breaking are well described in the simulation. Although the jet of first breaking hits a little faster ($t = 0.34$ s) in simulation than that of the experiment result, the simulation results generally show a good agreement with the experimental result.
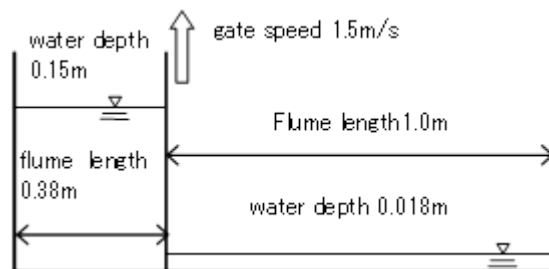

Figure 2. Initial condition of dam-break experiment conducted by Janosi et al. (2004)
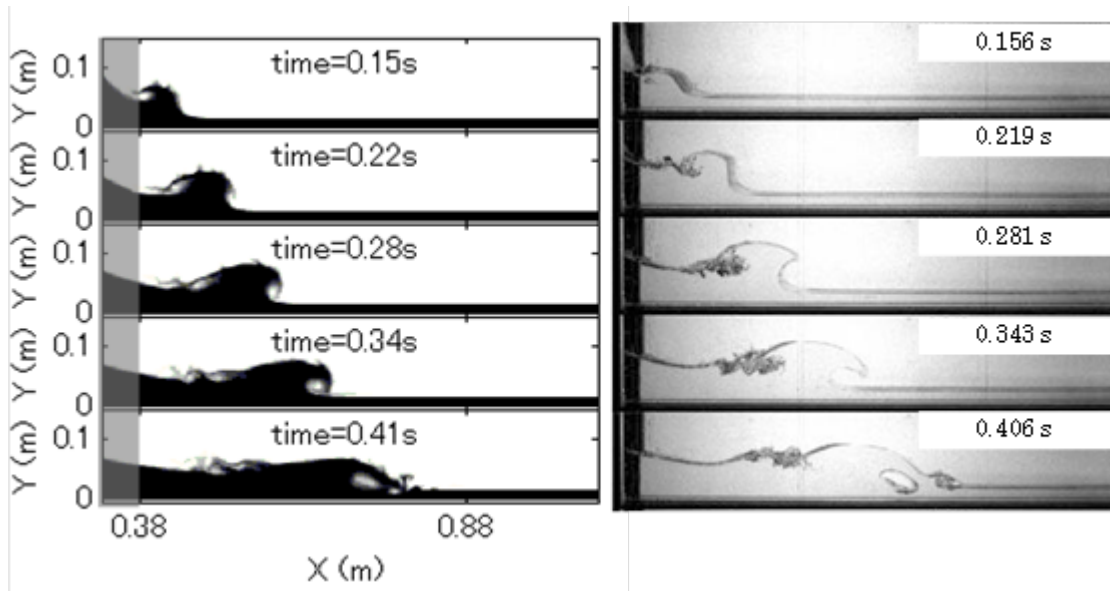
Figure 3.   A comparison of dam breaks between the numerical simulation and an experimental result by Janosi et al. (2004). Simulation results are on left side and experiments on right side.

Bore Wave Propagation

To verify simulation for bubbly flow calculations, a simulation result is compared with an experimental result of bore wave propagation with entrained air and babbles. A single bore wave is generated in a small wave flume which consists of a reservoir and a downstream part separated by a gate opened manually. The reservoir is 0.3 m in width and 1.43 m in length and water depth is set to be 0.33 m. At downstream part, the initial water depth is set to be 0.1 m. Bubbly flow field at 1.5 m downstream from the gate is compared.

In Fig. 4, photographs of the experiment and simulation results at 1.11, 1.31 and 1.51 s after the gate open are shown. In this case, a gate opening process which actually takes a certain fraction of second is not considered in the simulation. In the figure, bubbly flow is described as distributions of density function values between 0.0 and 1.0, and bubble entrainments are shown as areas with low density values. The simulation results give distributions of density function values at one *x-z* section, whereas the photos show superposition of phenomena in the *y*-direction. Thus it is difficult to compare these results quantitatively, but bubbly flow areas and low density function value areas show good agreement.

With propagation of the bore wave, an area with intermediate density at the nose of wave propagates faster than the wave propagation, which is not actually observed in the front of a bore wave. The reason may be the intermediate density area works as a buffer in the calculation. The impact of front edge of plunging water is not appropriately described in the calculation and the following movement of water is not properly simulated. Another point is that it is not clear whether the separation rate of air and water for bubbly flow introduced in the previous section is appropriate or not. The separation speed might be one of causes of the problem.

Wave Breaking of Solitary Wave on Slope

Two-dimensional wave breaking simulation of a single wave on a 1/20 constant slope is shown in Fig. 5. Height and width of a calculation grid are 0.01 m and 0.01 m. A sine wave is generated at the left side of the calculation region where the water depth is 0.2 m. Wave shoaling and breaking are calculated and distributions of the density function values at 1.11, 1.61 and 2.11 s after the wave generation are shown as contour maps in the figure.

A snapshot at a wave shoaling phase is shown in the top panel. Although the wave is not broken yet, an area with intermediate density value can be found at the front of wave. In the second panel, wave plunging is calculated. The area with intermediate density is expanded and the density value at the wave

crest shows rather small value. In the third panel, the wave after breaking is given. The intermediate density area now forms a bore wave.
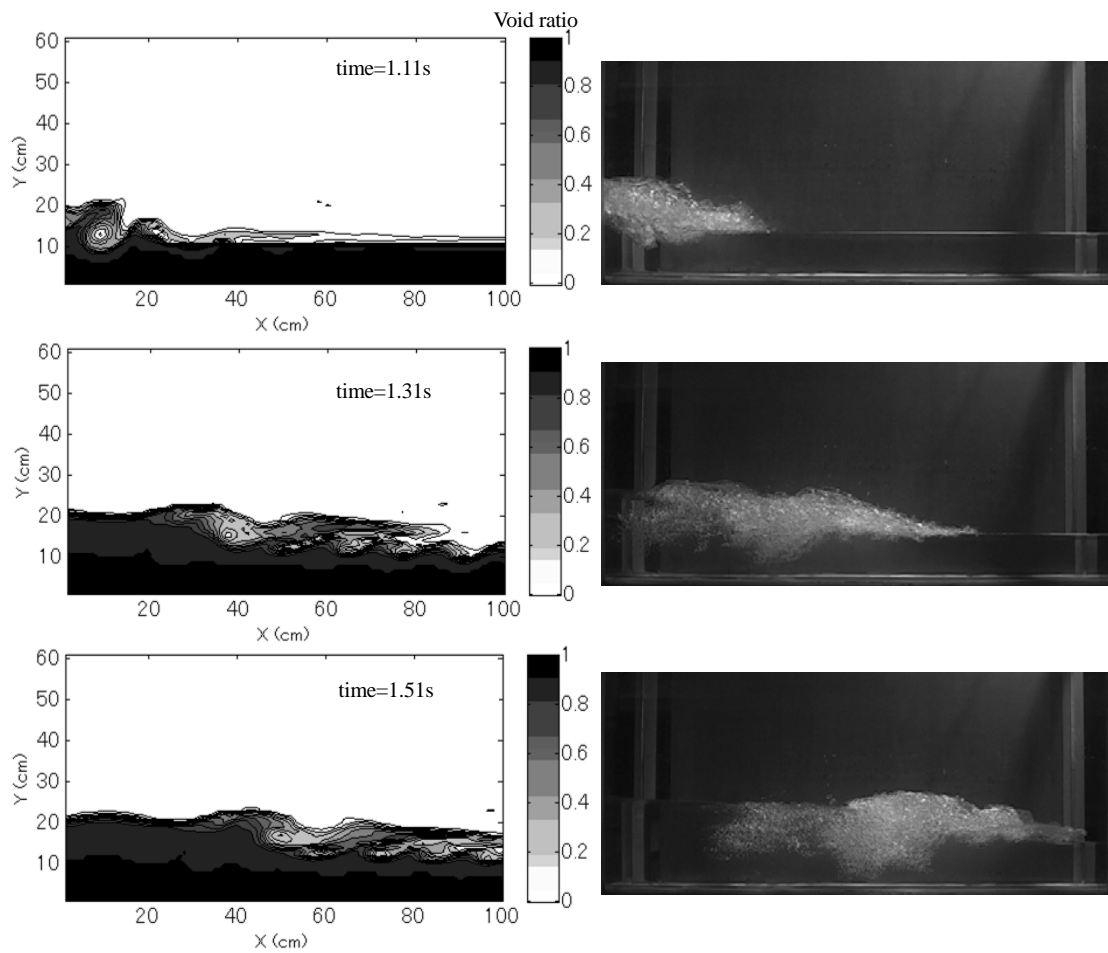

Figure 4. Bore wave propagation with bubbly flow. Simulation results on left side and photos of experiment on right side.
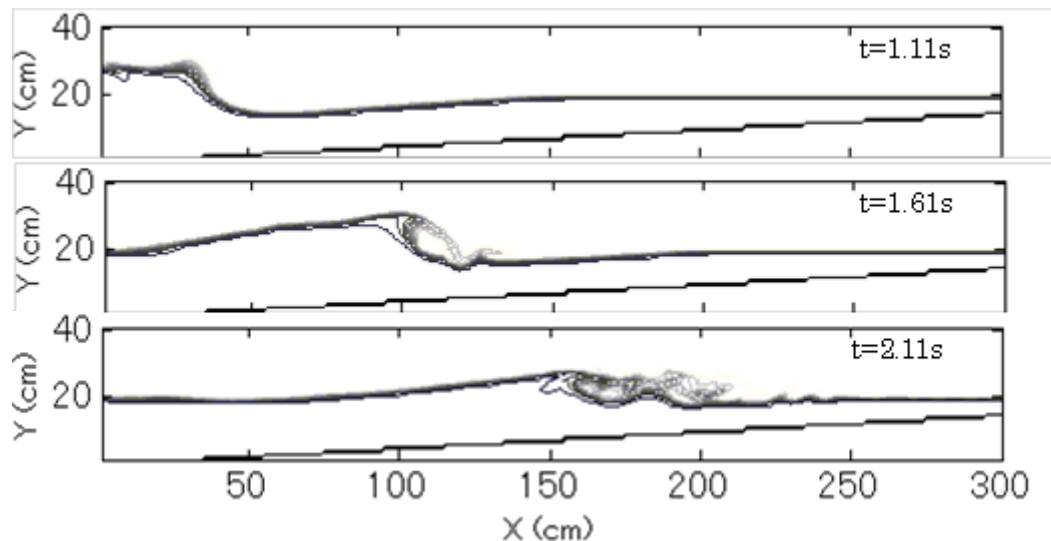

Figure 5. Wave breaking simulation of a single wave on slope

CONCLUSIONS

In order to simulate wave breaking by direct simulation without large computational load, a uni-phase flow model in which density of fluid is given by a value between that of air and water was developed. In the model, Navier-Stokes equation described in terms of momentum advection is employed as the governing equation to minimize the momentum conservation error as well as the mass conservation error for flows with large density change. Simulation results are compared with experimental results. Though effects of flow created by bubbles smaller than a grid is not well reflected to larger flow patterns, simulation results show reasonable agreement with experimental results for a dam-break, bore propagation and wave breaking on a slope.

REFERENCES

Gong, X., S. Takagi, H. Huang, and Y. Matsumoto, 2007. A numerical study of mass transfer of ozone dissolution in bubble plumes with an Euler-Lagrange method, *Chemical Engineering Science*, 62, pp. 1081-1093.

Janosi, I. M., D. Jan, K. G. Szabo and T. Tel. 2004. Turbulent drag reduction in dam-break flows, *Experiments in Fluids,* 37, pp.219-229

Kawasaki, K., 1999. Numerical simulation of breaking and post-breaking wave deformation process around a submerged breakwater, *Coastal Engineering Journal*, Vol.41,Nos.3&4, pp.201-223.

Lubin, P., S. Vincent, S. Abadie, J. Caltagirone, 2006. Three-dimensional large eddy simulation of air entrainment under plunging breaking waves, *Coastal Engineering*, 53, pp. 631-655.

Ma, D., M. Liu, Y. Zu. and C. Tang, 2012. Two-dimensional volume of fluid simulation studies on single bubble formation and dynamics in bubble column, *Chamical Engineering Science*, 72, pp.61-77.

Miyata, H., A. Kanai, T. Kawamura and J. Park, 1996. Numerical simulation of three-dimensional breaking waves, *Journal of Marine Science and Technology*, 1:183-197.

Mori, N. and S Kakuno, 2008. Aeration and bubble measurements of coastal breaking waves, *Fluid Dynamics Research*, Volume 40, Issues 7-8, pp. 616-626.

Mutsuda, H. and T. Yasuda, 2000. Numerical Simulation of turbulent air-water mixing layer within surf-zone, Proceedings of 27th Int. Conf. on Coastal Eng., Sydney, pp.755-768.

Watanabe, Y., H. Saeki, 1999. Three-dimensional large eddy simulation of breaking waves, *Coastal Engineering Journal*, 41, 3-4, pp. 281-301.

Watanabe Y., Y. Niida, A. Saruwatari and H. Saeki, 2009. Stochastic models of entrainment and advection of air bubbles under jets, *Journal of Coastal Engineering*, JSCE, 56, pp. 6-10. (in Japanese)

Yabe, T., M. H. Mizoe, K. Takizawa, H. Moriki, H. Im and Y. Ogata, 2004. Higher-order schemes with CIP method and adaptive Soroban grid towards mesh-free scheme, *Journal of Computational Physics*, 194, pp. 57-77.