Numerical Simulation and Experiment on Dam Break Problem

Changhong Hu[*] and Makoto Sueyoshi

Research Institute for Applied Mechanics, Kyushu University，
Kasuga Fukuoka 816-8580, Japan

Abstract: In this paper, two novel numerical computation methods are introduced which have been recently developed at Research Institute for Applied Mechanics (RIAM), Kyushu University, for strongly nonlinear wave-body interaction problems, such as ship motions in rough seas and resulting green-water impact on deck. The first method is the CIP-based Cartesian grid method, in which the free surface flow is treated as a multi-phase flow which is solved using a Cartesian grid. The second method is the MPS method, which is a so-called particle method and hence no grid is used. The features and calculation procedures of these numerical methods are described. One validation computation against a newly conducted experiment on a dam break problem, which is also described in this paper, is presented.

Keywords: CIP method; MPS method; dam break experiment

Article ID: 1671-9433(2010)02-0109-06

1 Introduction

This research aims computation of strongly nonlinear fluid structure interaction (FSI), which has drawn much attention from CFD researchers due to its increasing industrial needs in recent years. We have been developing a number of CFD techniques in past ten years; among those the most promising methods are the CIP (Constrained Interpolation Profile, Yabe, *et al*., 2001) based Cartesian grid method and the MPS (Moving Particle Semi-implicit, Koshizuka and Oka, 1996) method. The main purpose of this paper is to introduce these two methods.

The CIP based Cartesian grid has been developed by Hu and his colleagues for two dimensional problems (Hu and Kashiwagi, 2004, 2009) and three dimensional problem (Hu *et al.*, 2006a, 2006b, 2007, 2008). The wave-body interaction problem is treated as a multi-phase flow problem and solved numerically in a simple Cartesian grid, which does not depend on the free surface and the body boundary. Robust and efficient computations can be achieved easily by this method for the strongly nonlinear problems. In general, the method has the main advantage of its simple scheme and the main disadvantage of low grid resolution in the inner interfaces. To increase the computation accuracy in the inner interfaces, CIP method is applied as the base scheme to the flow solver.

Development of a MPS method for strongly nonlinear free-surface problems in ocean engineering was first carried out by Sueyoshi and Naito (2003, 2004). These days many other researchers seem to be working with this method and the number of related papers is increasing. In general, the MPS has the significant merits such as no numerical diffusion and perfect conservation of mass. Further, the MPS scheme is relatively robust and simple. On the other hand, demerits of the MPS method are known as heavy computation load and pressure oscillation. The researchers on MPS developments are concentrated on overcoming these demerits (Sueyoshi *et al*., 2008).

Another purpose of this research is to present a newly conducted high-quality experiment on a dam break problem for validation of those numerical methods. This kind of experiments is well documented and has been widely used for validation of free surface CFD methods (Hirt and Nichols, 1981; Hu and Kashiwagi, 2004). However, we are not satisfied with those published experiments for two reasons: (1) the experiments are all small-scale and most of them were conducted several decades ago; (2) There is no information about the partition plate (dam) movement in the experiments. The motivation of the new experiment is to conduct a larger-scale experiment with a modern measurement system to obtain detailed information including the motion of the partition plate. In this paper, the details about the experiment are explained. Corresponding numerical simulations by the CIP method and the MPS method are carried out and comparisons between the measurements and numerical simulations are made.

2 Numerical methods

2.1 CIP based Cartesian grid method

In the CIP based Cartesian grid method, the dam break

problem is treated as a two-phase fluid problem. The fluid is assumed incompressible and viscous. The CIP method is adopted as the flow solver. A fixed Cartesian grid that covers the whole computation domain is used. The free surface is considered as an inner interface that is calculated by an interface capturing method.

In the flow solver, a viscous and incompressible flow is considered. The governing equations are as follows:

$$\frac{\partial u_i}{\partial x_i} = 0 \qquad (1)$$

$$\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = -\frac{1}{\rho}\frac{\partial p}{\partial x_i} + \frac{1}{\rho}\frac{\partial}{\partial x_j}\left(2\mu S_{ij}\right) + f_i \qquad (2)$$

where $S_{ij} = \left(\partial u_i/\partial x_j + \partial u_j/\partial x_i\right)/2$. f_i stands for the body force, such as the gravity force, etc. Time evaluation of Eq. (2) is performed by a fractional step method in which the equation is divided into an advection step and two non-advection steps.

1) Advection step ($u^n \rightarrow u^*$) to calculate the advection.
2) First non-advection step ($u^* \rightarrow u^{**}$) to calculate the diffusion and so on.
3) Second non-advection step ($u^{**} \rightarrow u^{n+1}$) to treat the velocity-pressure coupling.

In the advection calculation, the CIP scheme is used. The basic idea of the CIP scheme is to construct the profile of a function within a computational cell by using a polynomial and to preserve it during the advection. This can be accomplished by using some supplementary constraints, such as the spatial gradient of the function, in addition to the values at grid points.

Therefore the advection phase includes the solution of two equations of the following kind:

$$\left.\begin{array}{l} \dfrac{\partial q}{\partial t} + u_j \dfrac{\partial q}{\partial x_j} = 0 \\[2mm] \dfrac{\partial q_\xi}{\partial t} + u_j \dfrac{\partial q_\xi}{\partial x_j} = -q_\xi \dfrac{\partial u_j}{\partial x_j} \end{array}\right\} \qquad (3)$$

where $q \equiv u_i$, $q_\xi = \partial q/\partial \xi$ and $\xi = x_1, x_2, x_3$. The calculation of the right hand side term in the second equation can be included in the first non-advection step.

Taking 1-D case as an example, an interpolation function using the cubic polynomial can be constructed to approximate the profile around a grid point x_m as follows:

$$Q_m(\eta) = a_m \eta^3 + b_m \eta^2 + c_m \eta + d_m \qquad (4)$$

Here $\eta = x - x_m$ and x is considered as a point located in the

upwind computational cell in terms of x_m, e.g., $[x_{m-1}, x_m]$ for $u_m > 0$. The four unknown coefficients in Eq. (4) can be determined using q_m, q_{m-1}, $(q_x)_m$, and $(q_x)_{m-1}$.

Once the interpolation function is constructed as described above, a semi-Lagrangian procedure is applied for the time evolution of Eq. (4) as follows:

$$\left.\begin{array}{l} q_m^* = Q_m^n(x_m - u_m \Delta t) \\[2mm] q_{xm}^* = \partial Q_m^n(x_m - u_m \Delta t)/\partial x \end{array}\right\} \qquad (5)$$

Here the superscript 'n' denotes the values at current time level and the superscript '$*$' denotes the values after advection calculation.

The velocity-pressure coupling is treated in the second non-advection step calculation, in which the following Poisson equation is used.

$$\frac{\partial}{\partial x_i}\left(\frac{1}{\rho}\frac{\partial p^{n+1}}{\partial x_i}\right) = \frac{1}{\Delta t}\frac{\partial u_i^{**}}{\partial x_i} \qquad (6)$$

where $\Delta t = t^{n+1} - t^n$ stands for the size of one time step. Eq.(6) is assumed valid for liquid and gas phase. Solution of it provides the pressure distribution in the whole computation domain. In this treatment, the boundary condition for pressure at the interface between different phases is not required.

In the dam breaking simulation, two density functions, ϕ_1 and ϕ_2, are defined to denote liquid and gas phase, respectively. The free surface is treated as an immersed interface.

The free surface can be determined by numerical solution of the equation for ϕ_1.

$$\frac{\partial \phi_1}{\partial t} + u_j \frac{\partial \phi_1}{\partial x_j} = 0 \qquad (7)$$

Several schemes have been tested in our code, such as CIP-CSL3 (Kishev *et al.*, 2006). Recently the THINC scheme (Xiao *et al.*, 2005), which is a conservative and smearing-less interface-capturing scheme, has been implemented to compute the time evaluation of ϕ_1 in the CIP based Cartesian grid method.

2.2 MPS method

The second numerical method used for the dam breaking simulation is the MPS method. Application and improvement of MPS method to ship hydrodynamic research was first done by the second author (Sueyoshi and Naito, 2003). The MPS method is a particle method which has the following features:

- Gridless Lagrangian method, and thus
- No numerical diffusion and perfect conservation of mass.
- Robust and simple because of no necessity of the geometric connectivity among particles.

Here by numbering the particle with a subscript i, the vectors of position and velocity of a particle can be denoted by $\boldsymbol{r}_i$ and $\boldsymbol{u}_i$. The pressure exerted by the fluid at $\boldsymbol{r}_i$ is denoted by p_i. From a Lagrangian viewpoint, they can be written as

$$\boldsymbol{u}_i = \boldsymbol{u}(t, \boldsymbol{r}_i), \quad p_i = p(t, \boldsymbol{r}_i) \tag{8}$$

In the present MPS method, only the water is considered in the computation, i.e., a single-phase computation will be conducted. The water is treated as incompressible. Thus the density of water ρ is constant from the continuity equation. The Navier-Stokes equations governing the fluid motion are described as

$$\frac{\mathrm{D}\boldsymbol{u}}{\mathrm{D}t} = -\frac{1}{\rho}\nabla p + \nu\nabla^2\boldsymbol{u} + \boldsymbol{f} \tag{9}$$

In order to solve Eq. (9) numerically, discrete models for the first- and second-order differential operators need to be introduced. In MPS method the particle interaction models are used. The first-order differential operator, the gradient of a scalar quantity ϕ, at the i-th particle is described as

$$\nabla\phi_i = \frac{d}{n^0}\sum_{j\neq i}^{N} w(r_{ij})\frac{\phi_j - \phi_i}{r_{ij}}\frac{\boldsymbol{r}_j - \boldsymbol{r}_i}{r_{ij}} \tag{10}$$

where $r_{ij} = |\boldsymbol{r}_i - \boldsymbol{r}_j|$ is the distance between particles i and j, d is the number of dimensions of the space, and $w(r_{ij})$ is the weight function in terms of the distance r_{ij}. The following weight function is commonly used:

$$w(r) = \begin{cases} \dfrac{r_0}{r} - 1 & \text{for } r \leq r_0 \\ 0 & \text{for } r > r_0 \end{cases} \tag{11}$$

Here r_0 is the cut-off radius which is constant and related to the influencing area in the particle interaction model.

The particle number density is defined as

$$n_i = \sum_{j\neq i}^{N} w(r_{ij}) \equiv n^0 \tag{12}$$

In the equation n^0 must be constant, because the fluid density is uniform in an incompressible fluid.

The second-order differential operator, the diffusion of a quantity ϕ, at the i th particle is modeled as

$$\nabla^2\phi_i = \frac{2d}{\lambda}\sum_{j\neq i}^{N} w(r_{ij})\left(\phi_j - \phi_i\right) \tag{13}$$

where λ is a parameter to adjust a distributed quantity to the analytical result and given by

$$\lambda = \sum_{j\neq i}^{N} r_{ij}^2 w(r_{ij}) \tag{14}$$

In the MPS method the pressure is solved by the following Poisson's equation.

$$\nabla^2 p_i^{n+1} = -\frac{\rho}{\Delta t^2}\frac{(n_i^* - n^0)}{n^0} \tag{15}$$

3 Experiment

With the CFD developments, a series of experiments were also conducted for validation purpose (Hu and Kashiwagi, 2004, 2006a). In this research, we consider a dam break experiment. A new experimental setup was built in our laboratory as shown in Fig.1. The water tank with a removable partition plate is used and the dimension is shown in Fig.2. In the experiment, at first the left compartment is filled with water to a certain level, then the partition plate is removed at a high speed. The flow motion is recorded by a high-speed digital video camera system. The speed of the water front as well as the speed of the partition plate was then obtained by analysis of the video images. The experiments were repeated more than 10 times and it is found that the repeatability is good.
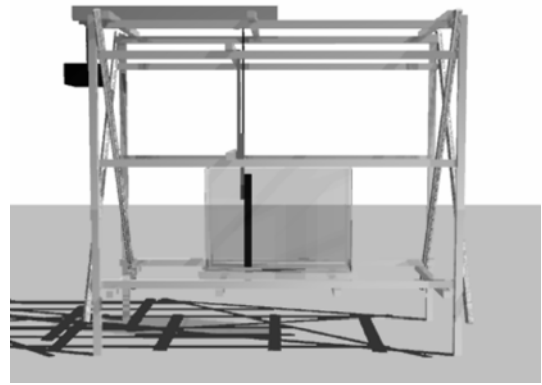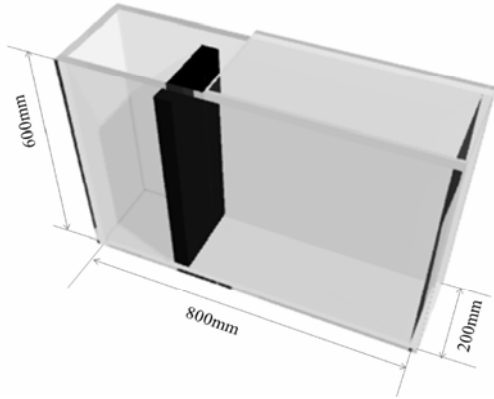

Fig.1 The experimental setup
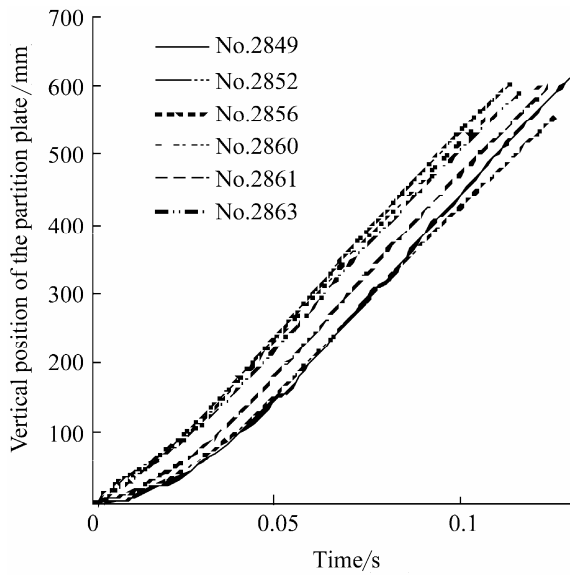
Fig.2 The water tank

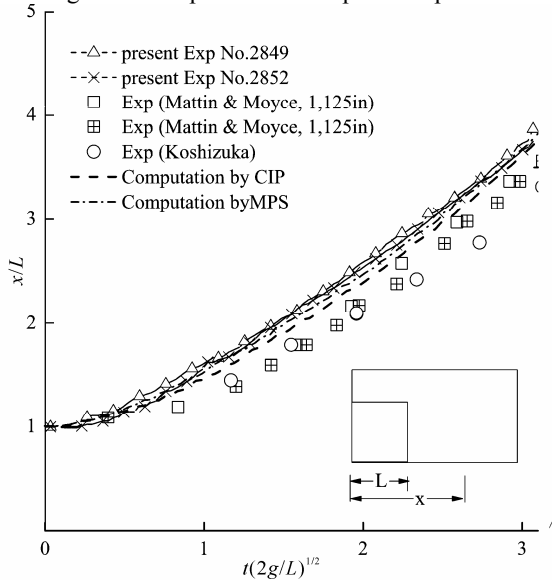
Fig.3 Vertical position of the partition plate

Fig.4 Comparison of water front speed

The speed of the partition plate is considered as a factor

which may affect the later motion of the fluid. Fig.3 shows the measured vertical position of the partition plate tip. Results of six runs are plotted. The vertical speeds for the partition plate are then obtained from Fig.3, which can be used in the numerical simulation.

4 Comparable computations

To check the accuracy of the experiment and the performance of the two proposed numerical methods which have been described in the above, the water front speed measured in present experiment was compared with the other published experiments (Martin and Moyce, 1952; Koshizuka and Oka, 1996) and the two numerical simulations in Fig.4. Two measurements are plotted in the figure. The comparison is generally good while the three well documented experiments show slower speed. Such difference can be explained partly due to the tank scale, and partly due to the method of measurement. Further investigation on this issue is required. It is also needed to point out that although the removing speed of partition plate is different for the two experimental cases, the difference for the water front speed is trivial. It seems that if the partition plate is removed at sufficiently high speed, its effect can be neglected.

A comparison of free surface variation is shown in Fig.5. The experimental pictures are taken from the video images. In the present numerical results, the motion of the partition plate is considered for the MPS computation, but not considered for the CIP computation. It can be seen that in the experiment the free surface becomes three dimensional after it heats the vertical wall. By comparison between the 2-D computations and the 3-D experiment, we can find that our numerical methods can simulate successfully the major structures of the real free surface profile.

5 Conclusions

Two novel CFD models, the CIP-based Cartesian grid method and the MPS method, are introduced in this paper for studying strongly nonlinear free surface problems. The methods are being developed to be both robust and accurate even for a case where large deformation of the fluid occurs, such as wave breaking, splash, air entrainment, and so on. A new experimental setup has been constructed for high-quality experiment on dam breaking. The motion of the free surface as well as the motion of partition plate can be accurately measured for the purpose of CFD model validation. A comparison of numerical results with the experiment and numerical results is carried out.
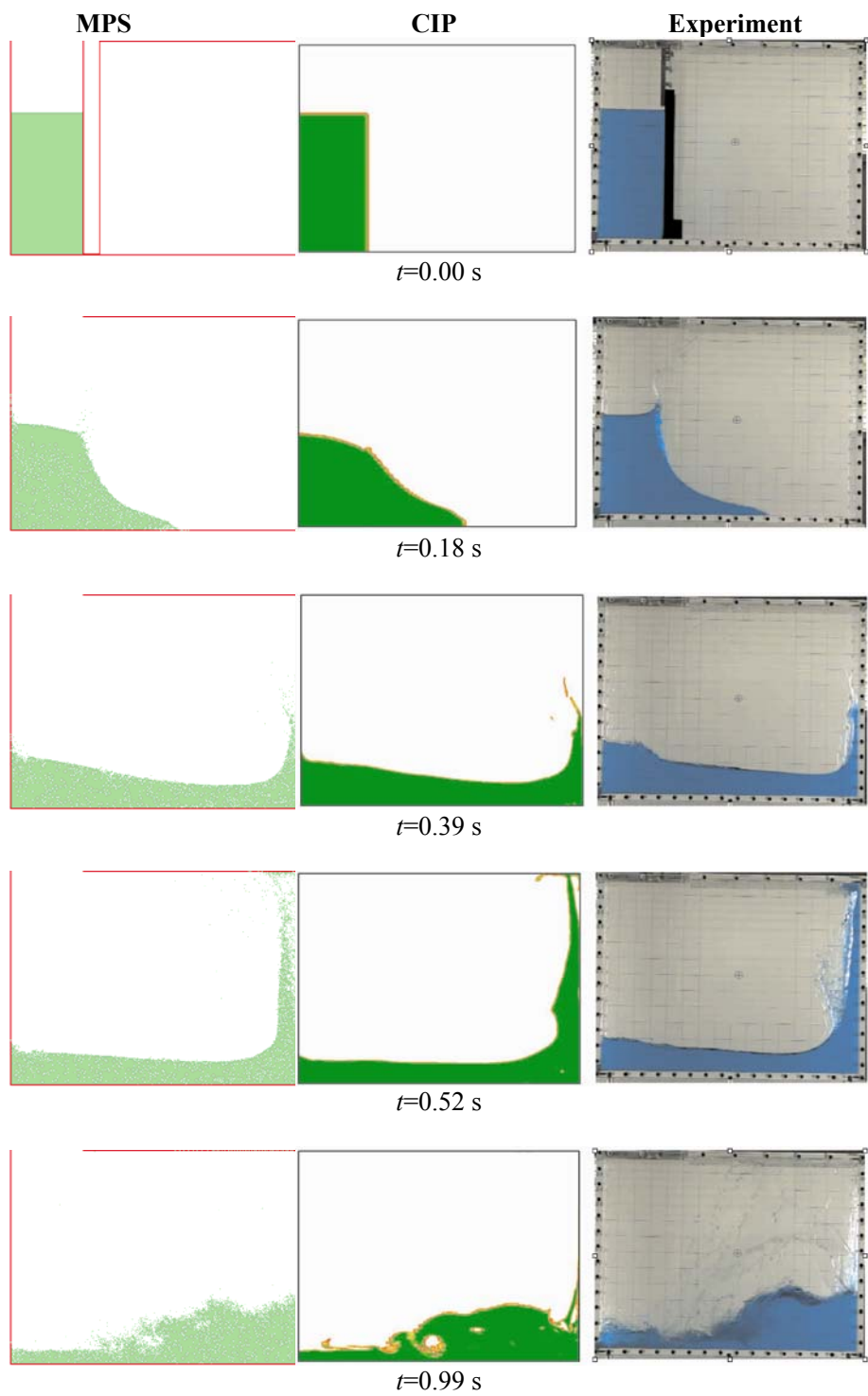
Acknowledgement

Fig.5 Comparison between experiment and computation

References

Hirt CW, Nichols BD (1981). Volume of fluid (VOF) methods for the dynamic of free boundaries. *Comput. Phys*，**39**，201-225.

Hu C, Kashiwagi M (2004), A CIP-Based method for numerical simulations of violent free surface flows. *Marine Science and Technology*, **9**(4), 143-157.

Hu C, Kashiwagi M (2006a). Validation of CIP-Based method for strongly nonlinear wave-body interactions. Proc. *26th Symposium on Naval Hydrodynamics*, Rome, **4**, 247-258.

Hu C, Kishev Z, Kashiwagi M, Sueyoshi M, Faltinsen OM (2006b). Application of CIP method for sStrongly nonlinear marine hydrodynamics. *Ship Technology Research*, **53**(2), 74-87.

Hu C, Kashiwagi M (2007). Numerical and experimental studies on three-dimensional water on deck with a modified wigley model. *Proc. 9th Numerical Ship Hydrodynamics*, Ann Arbor, Michigan, **1**, 159-169.

Hu C, Kashiwagi M, Sueyoshi M (2008). Improvement towards high-resolution computation on strongly nonlinear wave-induced motions of an actual ship. *Proc. of 27th Symposium on Naval Hydrodynamics*, Soeul, Korea, **5-10**, 525-534.

Hu C, Kashiwagi M (2009). Two-dimensional numerical simulation and experiment on strongly nonlinear wave–body interactions. *Marine Science and Technology*, **9**, 143-157

Kishev Z, Hu C, Kashiwagi M (2006). Numerical simulation of violent sloshing by a CIP-Based method. *Marine Science and Technology*, **11**, (2), 111-122.

Koshizuka S, Oka Y (1996). Moving-Particle Semi-Implicit method for fragmentation of incompressible fluid. *Nuclear Science and Engineering*, **123**, 421-434.

Martin JC, Moyce WJ (1952). An experimental study of collapse of liquid columns on a rigid horizontal plane. Philos. *Trans. R. Soc.. London Ser. A*, **244**, 312-324.

Sueyoshi M, Naito S (2003). A numerical study of violent free surface problems with particle method for marine engineering. *Proc.8th Numerical Ship Hydrodynamics*, Busan, **2**, 330-339.

Sueyoshi M, Naito S (2004). A 3-D simulation of nonlinear fluid problem by particle method – Over One Million Particles Parallel Computing on PC Cluster –. *Kansai Soc. Nav. Arch*,. Japan, **241**, 133-142.

Sueyoshi M, Kashiwagi M, Naito S (2008). Numerical simulation of wave-induced nonlinear motions of a two-dimensional floating body by the moving particle semi-implicit method. *Marine Science and Technology*, **13**, 85-94.

Xiao F, Honma Y, Kono T (2005). A simple algebraic interface capturing scheme using hyperbolic tangent function. *Int. J. Numerical Methods in Fluids*, **48**, 1023-1040.

Yabe Y, Xiao F, Utsumi T (2001). The constrained interpolation profile method for multiphase analysis. *Comp Physics*, **169**, 556-569.

Changhong Hu was born in 1964. He is an associate professor of Kyushu University, Japan. His current research interests include strongly nonlinear free surface problems, fluid-structure interaction analysis, CFD in marine hydrodynamics, *etc*.

Makoto Sueyoshi was born in 1977. He is an assistant professor of Kyushu University, Japan. His current research interests include multi-phase computation, large-scale simulation technique, CFD in marine hydrodynamics, *etc*.