A CIP BASED NUMERICAL SIMULATION METHOD FOR EXTREME WAVE-BOBY INTERACTIONS

Changhong Hu and Masashi Kashiwagi
RIAM, Kyushu University
6-1 Kasuga-koen, Kasuga, Fukuoka 816-8580, JAPAN
E-mail: hu@riam.kyushu-u.ac.jp

INTRODUCTION

Numerical simulation of the floating body motion in rough seas is a challenging subject. The main difficulty is that the topology of free surface may be largely distorted or broken up, which makes it impossible to apply the conventional numerical method such as the potential-flow based method by BEM. As there is a growing interest in the extreme wave-body interactions such as slamming, water on deck, wave impact by the green water and capsizing due to large-amplitude waves, development of new CFD simulation methods for the seakeeping researches is therefore required. Recently several challenging works have been made by the finite difference method in which the free surface is treated by VOF method (Greco et al., 2002), and by the particle method (Sueyoshi and Naito, 2001). Some preliminary results have been shown, but still many difficulties should be cleared before performing an engineering computation. The particle method requires very large computation time and memory and it is difficult to make a three dimensional calculation for an engineering problem at reasonable cost presently. For VOF method, solving three-dimensional wave breaking problem seems not easy because of its complex algorithms to treat the free surface. It is therefore necessary to develop a new numerical simulation method, which are both simple and robust enough to perform an engineering computation of extreme wave-body interactions.

In this paper a new finite difference method is proposed. The numerical model that is under development in RIAM, Kyushu University is based on the CIP (Constrained Interpolation Profile) method and the CCUP (CIP-combined and unified procedure) method (Yabe and Wang, 1991, Yabe et al., 2001). The CIP method is a recently developed semi-Lagrangian scheme for multiphase analyses, which includes a compact upwind scheme with subcell resolution, a pressure-based algorithm that can treat liquid, gas and solid phases by one equation.

For the wave-body interaction problems, a typical computation will include the calculation of numerical wave tank (NWT) and the calculation of floating body motion in it. In the present numerical method, this problem is viewed as a multiphase problem, which has a liquid phase (water), a gas phase (air) and a solid phase (wavemaker and floating body). They are solved numerically in a fixed Cartesian grid system by a finite difference method, in which the interfaces between different phases are tracked by a Eulerian method with a density function. The most important features of the method are as follows:

1. Air is included in the computation and its compressibility is considered.
2. The pressure for the whole computation domain (include solid body) is calculated by one set of equations.
3. The motion of floating body is calculated by a Lagrangian method.

We will describe the numerical method in the following section, in which the above features will be explained. Then a couple of two-dimensional numerical results are presented. The first example is about a two dimensional numerical wave tank, in which the waves are generated by a plunger-type wavemaker. An artificial damping zone is set at the end of the numerical wave tank. Both of the generated wave profiles and the hydrodynamic forces acting on the wavemaker are compared with the experiments as well as the results by a Mixed Eulerian-Lagrangian method. The second example is a numerical simulation on wave-body interaction demonstrating the capability of the present numerical method, in which a box-type floating body with an upper structure is used.

NUMERICAL METHOD

Governing Equations

Assuming that there is no temperature variation with the problem, the hydrodynamic equations can be written as follows:

$$\frac{\partial \rho}{\partial t} + u_i \frac{\partial \rho}{\partial x_i} = -\rho \frac{\partial u_i}{\partial x_i} \tag{1}$$

$$\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = \frac{1}{\rho} \frac{\partial \sigma_{ij}}{\partial x_j} + F_i \tag{2}$$

$$\frac{\partial p}{\partial t} + u_i \frac{\partial p}{\partial x_i} = -\rho C_S^2 \frac{\partial u_i}{\partial x_i} \tag{3}$$

Where C_S is the sound speed, σ_{ij} is the total stress. For a Newtonian fluid, the total stress can be written as $\sigma_{ij} = -p\delta_{ij} + 2\mu S_{ij} - 2\mu\delta_{ij} S_{kk}/3$, where $S_{ij} = \left(\partial u_i/\partial x_j + \partial u_j/\partial x_i\right)/2$. The second term on the right-hand side of Eq. (2) denotes the body force, which includes gravity force, surface tension force, etc. According to the CCUP method the pressure can be solved by the following equation.

$$\frac{\partial}{\partial x_i}\left(\frac{1}{\rho}\frac{\partial p}{\partial x_i}\right) = \frac{p - p^n}{\rho C_s^2 \Delta t^2} + \frac{1}{\Delta t}\frac{\partial u_i^n}{\partial x_i} \tag{4}$$

Note that this equation is valid for fluid, gas and sold phases, i.e. by solving Eq. (4), the pressure in the whole computation domain can be

obtained. Thus, the boundary conditions for pressure at the interfaces between different phases are not necessary, and fast solver or parallel computing technique can be easily applied. This is a very important feature, because the calculation of Eq. (4) is generally the most computationally time-consuming part for this kind of numerical simulation. Another advantage of Eq. (4) is that it provides a very simple and robust way to compute hydrodynamic forces acting on moving bodies in a fixed grid system.

CIP Method

According to CCUP method, the governing equations (1) - (3) are solved by a fractional step approach, i.e., at each time level, the advection is solved first, then the diffusion is calculated, at last the pressure is solved by Eq. (4). The advection calculation is performed by the CIP method. The basic idea of the CIP method is that for a value *f*, not only the advection equation of *f*, but also the advection equation of its spatial gradient $q_i = \partial f / \partial x_i$ are calculated by a cubic interpolation method. Therefore, the following equations are used for the advection calculation.

$$\frac{\partial f}{\partial t} + u_i \frac{\partial f}{\partial x_i} = 0 \tag{5}$$

$$\frac{\partial q_i}{\partial t} + u_j \frac{\partial q_i}{\partial x_j} = -q_j \frac{\partial u_j}{\partial x_i} \tag{6}$$

For three-dimensional cases, three extra equations must be solved. However, as the gradients in each cell can be solved, and only the information (value and its spatial gradients) at the grid points within one computation cell is needed for the interpolation procedure, this scheme has both subcell resolution and compact structure. This feature is very important for a multiphase computation in which there are discontinuities or large gradients in the interfaces.

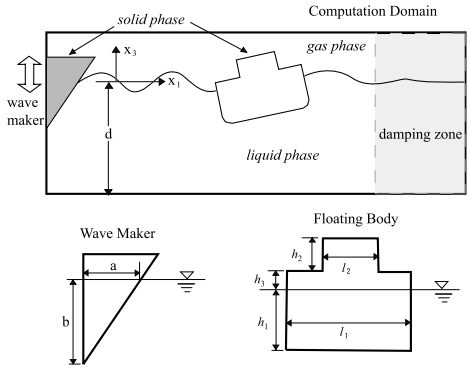


Fig.1 Schematic view of numerical wave tank.

Interface Tracking Method

The moving body boundary and the free surface boundary are distinguished by a density function ϕ_m , which is solved by the following equation.

$$\frac{\partial \phi_m}{\partial t} + u_i \frac{\partial \phi_m}{\partial x_i} = 0 \tag{7}$$

For the wave-body interaction problem shown in Fig. 1, *m =1*, *2*, *3* denotes liquid, gas, and solid phase, respectively.

The interface between gas and liquid, or the so-called free surface, is determined by solving Eq. (7) with the CIP scheme. Just like some other Eulerian interface tracking methods using density functions, the thickness of the interface will become a finite value due to the numerical diffusivity. However owning to the subcell resolution feature of the CIP scheme, the thickness will not grow larger and larger with the computation going on. Therefore for many practical problems, we have found that this degree of interface diffusion can be acceptable for a not very long time computation. A continuum surface force (CSF) model (Brackbill et al., 1992) is also involved in the present computation program to treat the surface tension, in which the surface tension is considered as a continuous, three dimensional effect across the interface that is identified by the density function. The surface tension force that can be included in the body force term F_i of Eq. (2) will have the following expression.

$$f_{Si} = -\frac{\sigma_S}{\rho} \nabla \cdot \left(\frac{\nabla \phi_1}{|\nabla \phi_1|} \right) \nabla \phi_1 \tag{8}$$

Here σ_S is the surface tension coefficient. No extra treatments are needed even when the interface is topologically distorted.

For calculating the motion of solid body boundary, as the floating body that we are interested in can be considered as a rigid body, we will not use Eq. (7) to obtain the density function for solid phase ϕ_3. Instead, as the geometry of the rigid body does not change with time, a Lagrangian method is developed to calculate ϕ_3 to obtain accurate solid body boundary positions without any numerical diffusion. We call this method as "geometric mapping method", for this method is used to map the geometry information of a moving body to a fixed Cartesian grid system. The basic idea for two-dimensional case is as follows. (The three-dimensional version is now under development.)

1. The two-dimensional body boundary is approximated by a series of straight line-segments: (p_k, p_{k+1}), $k = 1 \sim N$.
2. The coordinates for the end points (x_{pk}, z_{pk}) are calculated by the following equations.

$$x_{pk} = x_c + \left(x_{pk}^0 - x_c^0 \right) \cos\alpha - \left(z_{pk}^0 - z_c^0 \right) \sin\alpha \tag{9}$$

$$z_{pk} = z_c + \left(x_{pk}^0 - x_c^0 \right) \sin\alpha + \left(z_{pk}^0 - z_c^0 \right) \cos\alpha \tag{10}$$

   where (x_c, z_c) is the mass center of the floating body, α is the roll angle, the superscript 0 denotes the initial value. (x_c, z_c) and α are calculated in a Lagrangian way that will be described in the next section.
3. All of the intersection points (nodes) of line segments and grid lines are then calculated. For each computation cell, if there are more than 2 nodes, the cell is considered as a solid body boundary cell, and the area of the solid body in this cell is computed and then ϕ_3 in this cell is determined.

After the density function for all phases are calculated, the physical properties for each computation cell can be determined by the follow equation.

$$\lambda = \sum_{m=1}^{3} \phi_m \lambda_m \tag{11}$$

where λ denotes the viscosity, sound speed, etc.

Hydrodynamic Forces on the Floating Body

The hydrodynamic forces acting on the floating body can be obtained by integrating the pressure force and skin friction force along the body surface. If only the pressure is needed to consider, i.e., the skin friction is relatively small and can be neglected, the following Gauss theorem can be applied.

$$F_{di} = -\iint_S p n_i dS = -\iiint_V \frac{\partial p}{\partial x_i} dV \tag{12}$$

Where n_i is the unit normal vector. As the pressure in the whole computation domain can be calculated by Eq. (4), Eq. (12) provides a very simple and robust way to compute the hydrodynamic forces on the moving body in the fixed Cartesian grid system. For instance, for a two dimensional problem, the force and the moment acting on the floating body can be written as

$$F_x = -\iiint_\Omega \frac{\partial p}{\partial x}\phi_3 dV \ , \quad F_z = -\iiint_\Omega \frac{\partial p}{\partial z}\phi_3 dV \tag{13}$$

$$M_y = -\iiint_\Omega \left[(x - x_c)\frac{\partial p}{\partial z} - (z - z_c)\frac{\partial p}{\partial x} \right] \phi_3 dV \tag{14}$$

Where Ω denotes the whole computation domain. Then the position of the mass center and the roll angle can be obtained by the following equations.

$$\frac{d^2 x_c}{dt^2} = F_x \ , \quad \frac{d^2 z_c}{dt^2} = F_z \ , \quad \frac{d^2 \alpha}{dt^2} = M_y \tag{15}$$

Note that the skin friction force is not considered in Eq. (12) because the skin friction force is not continuous across the body surface. Therefore another approximate method is needed to develop for the problem in which the skin friction cannot be neglected.

Absorbing Boundary Condition for NWT

In order to perform simulations of numerical wave tank over long time in a finite computation domain, a non-reflection boundary condition is required at the downstream boundary. In this study, an artificial damping zone is placed at the downstream boundary ($x_1^s < x_1 < x_1^e$, $x_3^b < x_3 < x_3^t$), and an artificial damping force is added to the body force term of Eq. (2), which is expressed as follows.

$$f_{di} = \alpha \left(\frac{x_1 - x_1^s}{x_1^s - x_1^e} \right)^m \left(1 - \left| \frac{x_3 - x_3^f}{x_3^t - x_3^b} \right| \right)^n u_i \delta_{i3} \tag{16}$$

where x_1^s , x_1^e , x_3^b and x_3^t denote the positions of damping zone boundaries, x_3^f is the average free surface position. For constants in Eq. (16), $\alpha = 0.5/\Delta t$, $m = 4$, $n = 1$ are used for the computations shown in this paper according to several prior test calculations.

RESULTS

A couple of two-dimensional numerical examples will be shown for both validation and demonstration of the present method.

Numerical Wave Tank

The computation condition is chosen from an experiment (Kashiwagi, 1996). The characteristic length for the float of the wave maker is $a = 0.3776m$, the period of the wavemaker motion is $T\sqrt{g/a} = 7.343$. The computation grid and time step adopted are shown in Table 1. The comparison of free surface elevation at *x/a* = 9.629 with both the experiment and the non-linear BEM calculation by Kashiwagi, which is shown in Fig. 2, is perfect for this case.

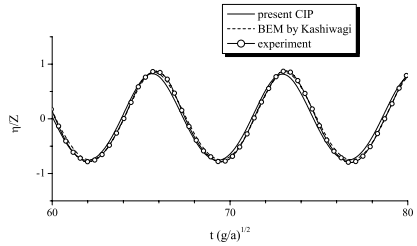


Fig.2 Comparison of free surface elevations at *x/a* = 9.629

Table 1 Computation condition for NWT

Wave maker	
geometry	a=0.0792m b/a=2.5
motion	$z(t) = Z\sin(2\pi t/T)$
amplitude	$Z/a = 0.6$
wave number	$Ka = \omega^2 a/g = 0.2, 0.4, 0.6, 0.8, 1.0$
water depth	d/a =7.6
Grid and Time Step	
grid number	300 (horizontal)×136 (vertical)
min grid spacing	$\Delta x/a = 0.02$ $\Delta z/a = 0.02$
time step	$\Delta t/T = 5\times10^{-4}$

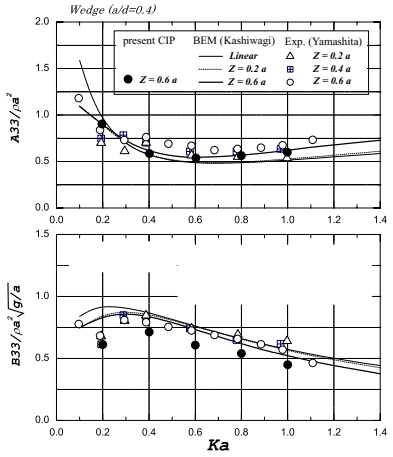


Fig.3 Comparison of first-order hydrodynamic forces

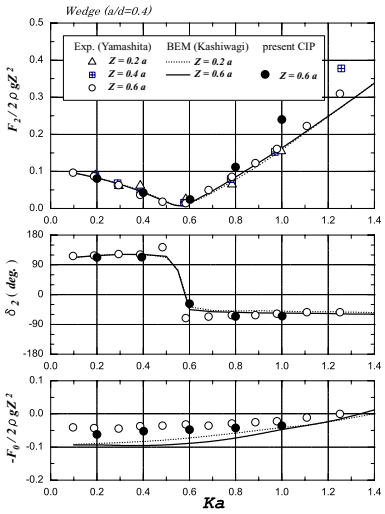
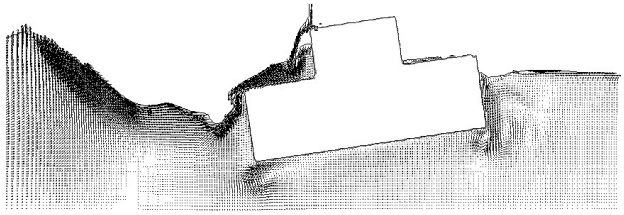


Fig.4 Comparison of second-order hydrodynamic forces

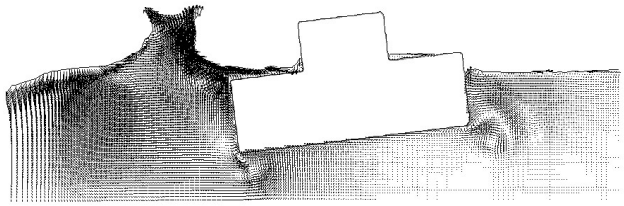
In Fig.3 and Fig.4 the hydrodynamic forces acting on the wavemaker are compared. The computation conditions are chosen from a forced oscillation experiment (Yamashita, 1977) with a wedge. The computation conditions are shown in Table 1. Except for the damping coefficients, the accuracy of present calculations is comparable to that of the non-linear BEM calculations.

2-D Wave-Body Interaction

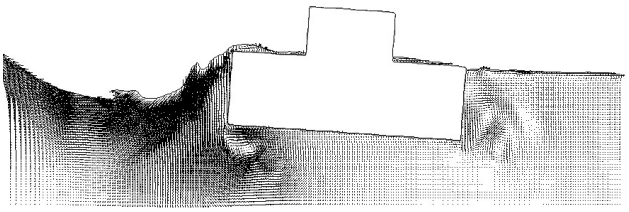
A two-dimensional numerical simulation on the interaction of floating body and non-linear waves is carried out. The general arrangement in the computation is shown in Fig.1. The computation condition for the wave tank is the same as that for Fig.3 and Fig.4. The parameters for the wavemaker are shown in Table 1, and the wave number of Ka = 0.4 is used. The dimensions for the floating body are: $l_1/a = 3.8$, $l_2/a = 1.4$, $h_1/a = 1.0$, $h_2/a = 0.8$, $h_3/a = 0.2$. The grid number used for the calculation is 500×145 .



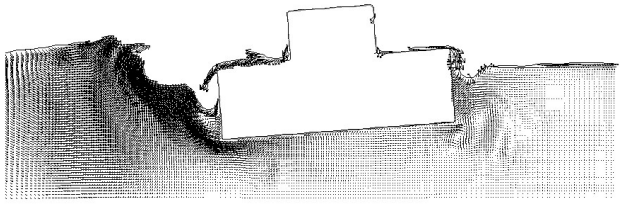
t/T=16.0



t/T=16.4



t/T=16.8



t/T=17.2

Fig.5 Interaction of floating body and non-linear waves

Fig.5 shows the calculated flow fields around the floating body at t/T=16.0, 16.4, 16.8, 17.2. It can be found that the extreme non-linear phenomena such as water on deck, wave breaking, vortex shedding are successfully simulated by the present numerical method.

CONCLUSIONS

This paper described a new numerical simulation method for the extreme wave-body interactions that is based on CIP and CCUP method. The improvement of the method as well as extension to three-dimension version is now in progress. Several 2-D numerical simulations have demonstrated the capability of this method, and it is expected that by further development this numerical method will become a powerful tool in seakeeping researches.

REFERENCES

Brackbill, JU, Kothe, DB, and Zemach, C (1992). "A Continum Method for Modeling Surface Tension", *J. Comput. Phys*., Vol 100, pp335-354

Greco, M, Faltinsen, OM and Landrini, M (2002). "Water Shipping on a Vessel in Head Waves," *Proc. of 24th Symp. On Naval Hydrodynamics*. Fukuoka, Japan, Vol 2, pp 40-52

Kashiwagi, M (1996). "Full-Nonlinear Simulations of Hydrodynamic Forces on a Heaving Two-Dimensional Body," *J. Soc. Nav. Arch. Japan*, Vol 180, pp 373-381.

Sueyoshi, M and Naito, S (2001). "A Study of Nonlinear Fluid Phenomena with Particle Method (part 1) Two-Dimensional Problems," (in Japanese) , *Journal of KSNAJ*, No 236, pp 191-198.

Yabe, T and Wang, PY (1991). "Unified numerical procedure for compressible and in-compressible fluid", *Journal of the Physical Society of Japan*, Vol 60, pp 2105-2108.

Yabe, T, Xiao, F and Utsumi, T (2001). "The Constrained Interpolation Profile Method for Multiphase Analysis", *J. Comput. Phys*., Vol 169, pp 556-593

Yamashita, S (1977). "Calculations of the Hydrodynamic Forces Acting Upon Thing Cylinders Oscillating Vertically with Large Amplitude", (in Japanese) , *J. Soc. Nav. Arch. Japan*, Vol 141, pp 61-70.