3-D Numerical Wave Tank by CIP based Cartesian Grid Method

Changhong Hu
Research Institute for Applied Mechanics, Kyushu University, Fukuoka, Japan
hu@riam.kyushu-u.ac.jp

1 Introduction

A Cartesian grid method with CIP (Constraint Interpolation Profile [1]) based flow solver has been developed by Hu and his colleagues for strongly nonlinear free surface problems, such as violent sloshing inside a liquid tank, slamming, water on deck, wave impact by green water, capsizing of ships or ocean structures, and so on.

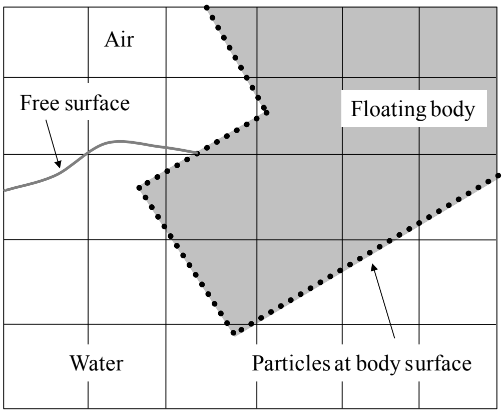

Fig.1 Multi-phase computation of wave-body interactions.

The wave-body interaction problem in our numerical model is treated as a multiphase problem as shown in Fig. 1, which includes a liquid phase (water), a gas phase (air) and a solid phase (floating body), which is solved numerically in a Cartesian grid that covers the whole computation domain. The free surface and the body boundary are treated as immersed interfaces. To recognize different phases we define a density function ϕ_m, in which m=1, 2, 3 denotes the liquid, gas, and solid phase, respectively. The density function for each computational cell has the relation of $\sum \phi_m = 1.0$ and follows the equation:

$$\frac{D\phi_m}{Dt} = 0 \qquad (1)$$

There are two kinds of the interface in the wave-body interaction problem: the gas-liquid interface (free surface) and the solid-fluid interface (body boundary) and different methods are used to calculate their motions.

To increase the computation accuracy in the inner interfaces, the CIP method, a high order upwind scheme with compact structures, is employed in our numerical method. The CIP method has been developed to have many variations for different purpose of computations [1]. For flow solver, we have applied and checked the performance of conventional CIP scheme, conservative CIP scheme, rational CIP scheme, in our numerical method. For interface capturing scheme, we have implemented and improved original CIP scheme, CIP-CSL3 scheme [2], and THINC scheme [3]. For calculation of floating body motion in the Cartesian grid, a weekly coupled procedure has been developed for our method, in which the fluid structure interaction is treated by using an immersed boundary method.

Such CIP based Cartesian grid method has been developed and improved for several years, to make it not only be able to handle complicated flow phenomena but also be highly efficient to perform three-dimensional simulations in an acceptable spatial and temporal resolution at reasonable cost. Development and validation of the CIP based Cartesian grid method for two-dimensional problems have been systematically carried out on violent free surface flows [4][5] and on strongly nonlinear wave-body interactions [6]. Experiments on those two-dimensional problems have been performed for validation of the numerical method. Good agreement has been obtained from the comparison between the numerical and the experimental results.

Our final goal is to develop a three-dimensional CFD code based on the CIP based Cartesian grid method that is capable of quantitatively predicting both global motions/loads and local impact loads for strongly nonlinear wave-ship interaction problems. Therefore our recent researches have been concentrated on the issues related to the three-dimensional numerical wave tank (3-D NWT)[7][8]. In this paper our 3-D NWT concept and some new improvements will be described. Numerical results by using the 3-D NWT will also be presented to demonstrate the performance of our three-dimensional code.

2 3-D NWT

The 2-D numerical simulation by the CIP based Cartesian grid method can be done by using a numerical wave tank that simulates a real wave channel as shown in Fig.2, in which the wave maker can be included in the computation. However, for a 3-D numerical simulation of wave-body interactions, the computation domain has to be reduced due to the restriction of available computer resources. In Fig.2, the concept of our 3-D NWT is shown, in which the computation domain is restricted to the neighborhood space of the ship. A Cartesian grid is used and the whole computation grid is moving with the advancing ship speed $\mathbf{U}_0$.

2.1 Flow Solver

For the flow solver, an unsteady, viscous and incompressible flow is considered in the computation. The governing equations are as follows:

$$\nabla \cdot \mathbf{u} = 0 \tag{2}$$

$$\frac{D\mathbf{u}}{Dt} = -\frac{1}{\rho}\nabla p + \frac{1}{\rho}\nabla \cdot \mathbf{T} + \mathbf{f} \tag{3}$$

where $D/Dt = \partial/\partial t + (\mathbf{u} - \mathbf{U}_0)\cdot\nabla$, $\mathbf{T}$ is the stress tensor. The last term on the right-hand side of Eq. (3) stands for the body force, such as the gravity force, etc. Time evaluation of Eq. (3) is performed by a fractional step method in which the equation is divided into an advection step and two non-advection steps. In the advection step, numerical calculation is done by the CIP scheme. The pressure is treated in a non-advection step calculation, in which the following Poisson equation is used.

$$\nabla \cdot \left(\frac{\nabla p}{\rho} \right) = \frac{1}{\Delta t} \nabla \cdot \mathbf{u} \tag{4}$$

Equation (4) is assumed valid for liquid, gas and solid phase. Solution of it gives the pressure distribution in the whole computation domain. The pressure distribution obtained inside the solid body is a fictitious one, which satisfies the divergence free

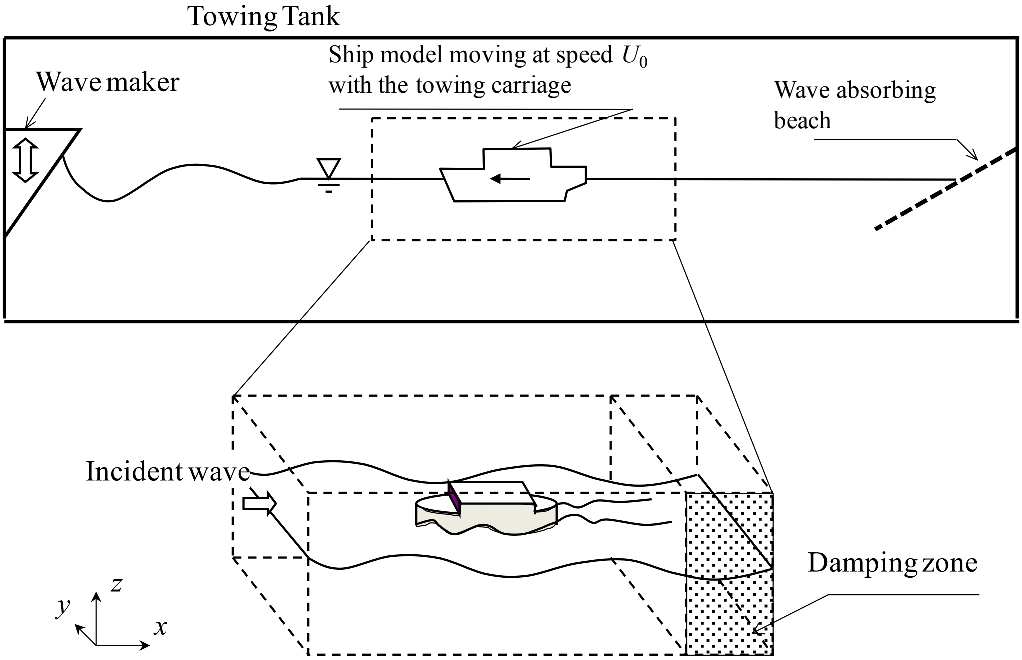

Fig.1 3-D NWT moving at speed U_0

condition of the velocity field. In this treatment, the boundary condition for pressure at the interface between different phases is not necessary, and a fast solver or parallel computing technique can be easily applied.

2.2 Interface Capturing Method

The motion of the free surface is calculated by a conservative interface capturing scheme, the THINC scheme. The following conservation form of the governing equation for the density function of fluid $\phi(\mathbf{x},t)$ is solved.

$$\frac{\partial \phi}{\partial t}+\nabla\cdot\left[\left(\mathbf{u}-\mathbf{U}_0\right)\phi\right]=\phi\nabla\cdot\mathbf{u} \tag{5}$$

Integrating this equation over a time interval $[t, t+\Delta t]$ and a cell $\left[x_{i-1/2,j,k}, x_{i+1/2,j,k};\quad y_{i,j-1/2,k}, y_{i,j+1/2,k};\quad z_{i,j,k-1/2}, z_{i,j,k+1/2}\right]$ gives:

$$\bar{\phi}_{i,j,k}^{n+1}=\bar{\phi}_{i,j,k}^{n}+\frac{1}{\Delta x_i \Delta y_j \Delta z_k}\left(g^x_{i-\frac{1}{2},j,k}-g^x_{i+\frac{1}{2},j,k}+g^y_{i,j-\frac{1}{2},k}-g^y_{i,j+\frac{1}{2},k}+g^z_{i,j,k-\frac{1}{2}}-g^z_{i,j,k+\frac{1}{2}}\right)+\bar{\phi}_{i,j,k}^{n}\left(\nabla\cdot\mathbf{u}\right)_{i,j,k}^{n} \tag{6}$$

where $\bar{\phi}_{i,j,k}$ is the cell averaged density function, g^x, g^y, g^z are the flux at the cell boundaries.

The flux is computed by a semi-Lagrangian scheme. A piecewise modified hyperbolic tangent function is used to approximate the profile of ϕ in the cell. Details about the interface capturing scheme can be found in [3] and [6].

2.3 6DoF Motion of Ship

The floating body is considered as a rigid body in the numerical model. When the hydrodynamic forces on the body surface are obtained by the flow solver, it is not difficult to calculate translational and rotational velocities at the gravity center of the rigid body. In order to treat large-amplitude ship motions, e.g., capsizing, the rotation of the body is solved by using quaternion representation.

The fluid structure interaction is solved in an explicit way:

- the floating body is solved by using the hydrodynamic forces on the body surface from the fluid solution
- the fluid field is solved in which the motion of the floating body is treated by using an immersed boundary method.

2.4 Boundary Condition for Outer Boundaries

On the in-flow boundary, velocities and free surface elevation, which are obtained by the Stokes 2nd order theory, are given at the boundary cells.

On the out-flow boundary, an artificial damping zone is placed at ($x^s < x < x^e$, $z^b < z < z^t$), and an artificial damping force is added to the body force term of Eq. (3), which is expressed as follows:

$$f_{di}=\alpha\left(\frac{x-x^s}{x^s-x^e}\right)^m\left(1-\left|\frac{z-z^f}{z^t-z^b}\right|\right)^n w \tag{8}$$

For other boundaries, slip boundary condition for velocities are used.

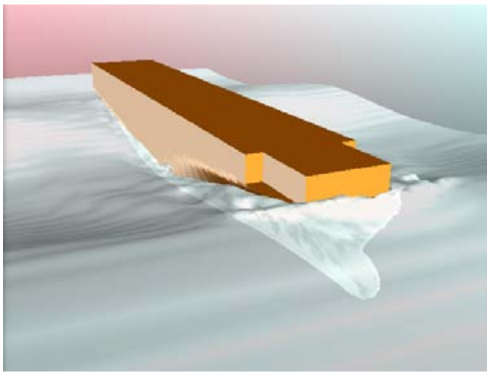

Fig.3 Strongly nonlinear wave-ship interaction computation

3 Numerical Results

As an example to demonstrate the performance of the proposed 3-D NWT, a numerical simulation result on a large container ship advancing in larges is presented in this section. The computation is performed according to an experimental case [9].

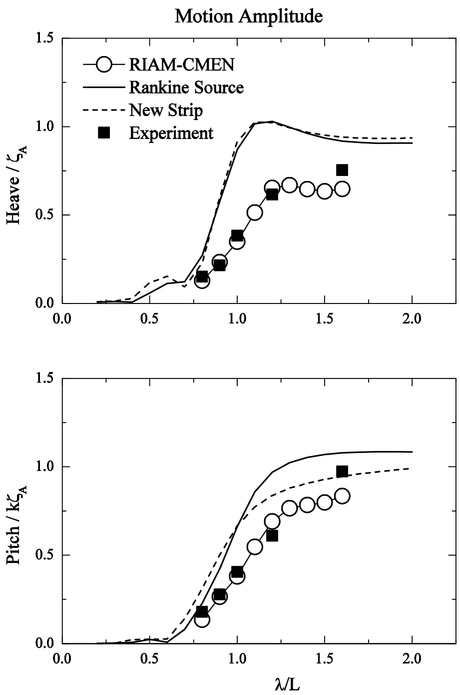

Fig.4 Comparison of ROA of ship motions

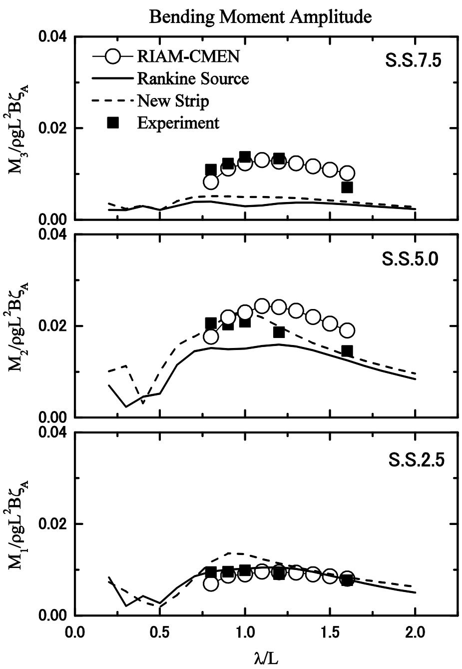

Fig.5 Comparison of ROA of vertical bending moments at the sections.

Fig. 3 is a computed snapshot to show the violent interaction between the ship and the free surface. The frequency response characteristics of the wave-induced motions are shown in Fig.4 and the vertical bending moment on three cross-sections of the ship are shown in Fig.5. The present numerical results are compared to the experiments and the results obtained by two conventional potential flow-theory based numerical methods [9]. It can be seen that by comparing to the experimental data, the accuracy of prediction by our 3-D NWT is much better than those obtained by the potential flow theory methods.

References

[1] Yabe, T., Xiao, F. and Utsumi, T. (2001) The Constrained Interpolation Profile Method for Multiphase Analysis, J. Comput. Phys., 169: 556-593

[2] Xiao, F. and Yabe, T. (2001) Completely conservative and oscillation-less semi-Lagrangian schemes for advection transportation, J. Comput. Phys., 170: 498-522

[3] Xiao, F., Ikebata, A. and Hasegawa, T. Numerical simulations of free interface fluids by a multi-integrated moment method, Computers and Structures 83 (2005) 409-423

[4] Hu, C. and Kashiwagi, M. (2004) A CIP-Based Method for Numerical Simulations of Violent Free Surface Flows, J. Marine Science and Technology, 9: 143-157

[5] Kishev, Z., Hu, C. and Kashiwagi, M. (2006), Numerical simulation of violent sloshing by a CIP-based method J. Marine Science and Technology, Vol. 11, pp. 111-122

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

[7] Hu, C. and Kashiwagi, M. (2007). Numerical and Experimental Studies on Three-Dimensional Water on Deck with a Modified Wigley Model, Proc. 9th Int. Conf. on Numerical Ship Hydrodynamics, Michigan, USA, V1, 159-169.

[8] Hu, C., Kashiwagi, M. and. 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, Oct. 5-10, 2008, pp. 525-534.

[9] Miyake, R., Mizukami, S., Ogawa, Y., Zhu, T., and Kumano A. (2004): Studies on Wave Loads Acting on a Large-Container Ship in Large Waves (in Japanese), Journal of SNAJ, Vol 195, pp 185-194.