Abstract for the 29th Intl Workshop on Water Waves and Floating Bodies, Osaka (Japan), March 30 – April 02, 2014

Prediction of Added Resistance in Short Waves by CFD Simulation

by Changhong Hu[1) *] Takashi Mikami[2)] and Koutaku Yamamoto[3)]

[1)] Research Institute for Applied Mechanics, Kyushu University, Fukuoka, Japan

E-mail: *hu@riam.kyushu-u.ac.jp*

[2)] Akishima Laboratories (Mitsui Zosen) Inc.

[3)] Mitsui Engineering & Shipbuilding Co.,ltd. (MES)

Highlights:

- CIP based Cartesian grid method has been developed and applied to predict added resistance in short waves which is mainly due to wave reflection and breaking at the bow.
- Grid convergence test shows that sufficient grid resolution is necessary for the accurate prediction of the added resistance by CFD simulation.

1. Introduction

For a large ship, most of the time at sea is in short wave condition. For very short waves ship motions are negligible, and the added resistance is mainly due to wave reflection and breaking at the bow. Theoretical methods have been proposed to predict the added resistance due to wave reflection in short waves, such as Faltinsen's asymptotic formula [1] and the semi-empirical formula by Fujii and Takahashi [2]. However, the bow shape above the still water line, i.e., bow flare, which is considered as an important factor in designing large merchant ships, cannot be considered by those methods. Towing tank experiment may be the only practical method to study the effect of the bow flare shape on the added resistance. However, the experiment is generally expensive and has the difficulty in short wave generation and hydrodynamic force measurement. In this study, as a collaboration research project with MES, we try to apply a CFD method to calculate the added resistance in short wave region by numerical simulation of wave-ship interaction in a numerical wave tank. In the computation, the wave reflection and breaking can be modeled with the consideration of the hull form above the still water line.

The CFD method used in the present study is an in-house code, which has been developed by the first author for more than ten years. For the purpose of predicting added resistance due to wave reflection and breaking at the bow, which is proportional to the square of the wave amplitude and therefore very sensitive to the grid resolution, new development and improvement have been carried out. In this extended abstract, the development and improvement of the CFD code are outlined at first, and then some recent obtained numerical results on a KVLCC model are presented. More detailed description on this study will be given at the workshop.

2. Numerical Method

The present CFD code is based on the CIP based Cartesian grid method [3]. There are four key features: (1) a Cartesian grid approach for multi-phase calculation; (2) CIP method [4] applied in the flow solver for water and air; (3) THINC (Tangent of Hyperbola for INterface Capturing) scheme [5] for capturing the free surface; and (4) an immersed boundary method for wave-body interaction treatment.

The wave-ship interaction problem is treated as a multi-phase flow problem that includes a liquid phase (water), a gas phase (air) and a solid phase (floating body). The whole computation domain, which moves at constant forward speed of the ship, is discretized by a fixed regular grid. To recognize different phases we define a density function ϕ_m, in which m=1, 2, 3 denotes liquid, gas, and solid phase, respectively. The density function in each computational cell satisfies $\sum\phi_m = 1.0$ and is calculated by the following equation:

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

where $D/Dt = \partial/\partial t + (\mathbf{u} - \mathbf{U}_0)\cdot\nabla$, $\mathbf{U}_0$ is the velocity of the grid.

The free surface is the interface between water and air, which can be sharply determined by numerical solution of Eq.(1) using THINC scheme. By such interface capturing method the violent variation of the free surface such as wave breaking can be simulated.

The ship is considered as a rigid body which is immersed in the Cartesian grid. A weekly coupling procedure has been developed to treat the wave-ship interaction. In each time step, first the flow field is solved in which the effect of the floating body is approximated by adding a body force to relevant computation cells. Next the hydrodynamic forces on the body surface are obtained and the motion of the body is calculated. In order to treat large-amplitude ship motions, the rotation of the body is solved by using quaternion representation.

The governing equations for the water-air multi-phase flow are the following incompressible continuity and Navier-Stokes equation.

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

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

where **T** is the stress tensor, **f** is the body force. A finite difference method is applied for numerical solution of those equations. Time evaluation of Eq. (3) is performed by a fractional step method in which the equation is divided into one advection step and two non-advection steps. In the advection step, CIP is applied as an upwind scheme.

The Cartesian grid method has the advantages of simplicity in coding and robustness in violent free surface calculation, but suffers from the difficulty in local grid refinement. Accurate prediction of the added resistance in short waves requires proper simulation of wave reflection and breaking at the bow vicinity. Two major developments have been made on the CFD code for this purpose. The first is the development of a panel representation method to define the body surface, as well as a set of interpolation schemes to impose the boundary conditions on the body surface. This new feature of the CFD code largely increases the calculation accuracy on the body surface. The second development is parallelization of the CFD code. In the parallelization development, 2-direction domain decomposition is used to partition the computation domain in which necessary routines have been added to the code under MPI standard. Non-blocking communication is used in the subroutines with large bulk and frequent data transmission to decrease the network transfer cost and increase parallelism of IO and CPU. The newly developed parallel version of the CFD code has been carefully validated and enables us to perform high resolution simulation using a grid with more than a hundred million grid points in our laboratory PC cluster system.

3. Results

In this study we chose a KVLCC model for testing the numerical method. To provide a benchmark for validation of the CFD code, an experiment has been conducted in the small towing tank (100m length 5m width and 2.65m depth) of Akishima Laboratories (Mitsui Zosen) Inc. on a ship model with the length of L=3.5m. The Froude number is Fn=0.144 and the wave height takes h/L=0.00906 and 0.0181.

3.1 Grid Convergence Test for a Motion Fixed Model

The performance of the numerical method in predicting the added resistance in short waves depends on how accurately the wave reflection and breaking at the bow vicinity are simulated. A grid convergence test is carried out for three short wave cases of $\lambda/L = 0.5, 0.6, 0.7$. In the computation, ship motion is fixed for that we want to simplify the condition and assume that the effect of the ship motion on the added resistance is negligible. Five grids are used and the details are shown in Table 1. Since the ship motion is fixed the grid points are concentrated in the vicinity of the bow. The computations are performed using the serial version of the CFD code in a personal computer (Pentium4 3.4G, Memory DDR 2G).

A comparison between the experimental photo and the CFD simulation is shown in Fig.1. The grid point number is 8M (8 million). Wave reflection and wave breaking are found in both the experiment and the numerical simulation. Small droplets due to wave breaking in the experimental photo cannot be reproduced by the CFD for insufficiency of the grid resolution. In Fig.2, the added resistance coefficients obtained from the CFD simulations with different grid resolution are compared to the experimental data and the result by the semi-empirical formula of Fujii and Takahashi. The convergence is found for the computation with grid point number more than 8M.

Table 1 Computation condition for grid convergence test

Ship motion		Restricted				
Wave length λ/L		0.5, 0.6, 0.7				
Wave height h/L		0.0181 (H=6m for full scale)				
Mesh Number	X	155	210	260	300	430
	Y	80	80	100	156	200
	Z	80	110	150	170	180
	Total	1 M	2 M	4 M	8 M	16 M
Minimum grid spacing	$\Delta x_{\min}/L$	0.010	0.005	0.004	0.003	0.002
	$\Delta y_{\min}/L$	0.005	0.005	0.004	0.003	0.002
	$\Delta z_{\min}/L$	0.005	0.002	0.002	0.0015	0.0015

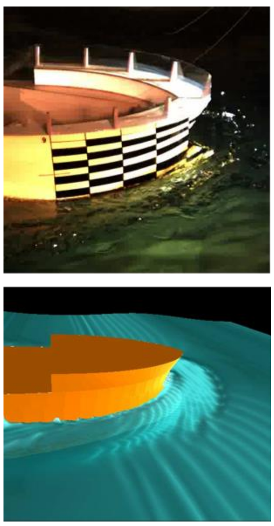


Fig.1 A comparison between experiment and CFD

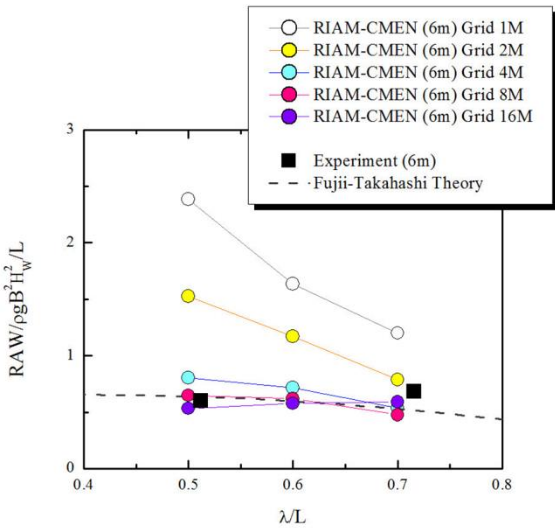
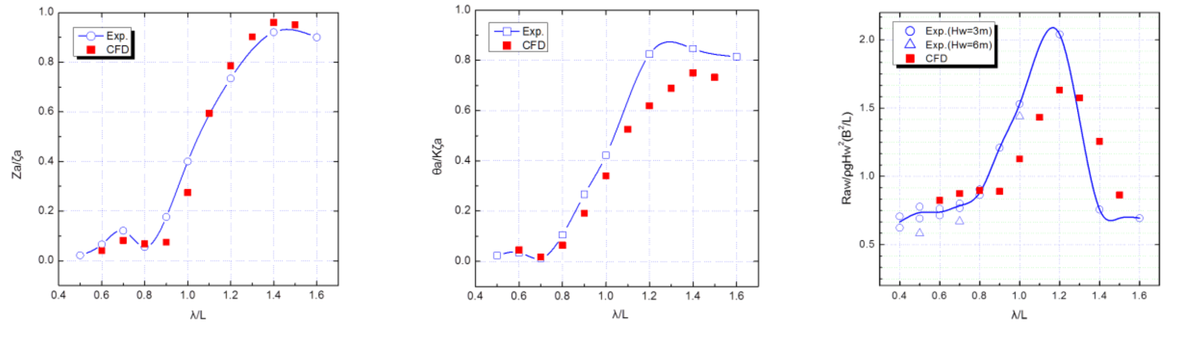


Fig.2 Grid convergence test on added resistance

3.2 Results for a Motion Free Case

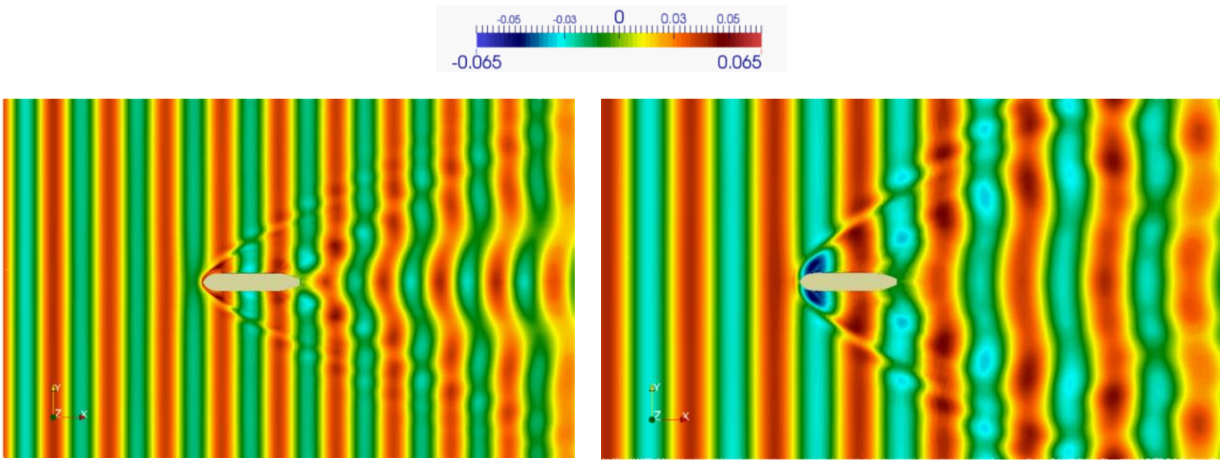
Numerical simulation on the experiment case is performed, in which the ship motion is considered. To achieve a sufficient grid resolution as discussed in the previous section, a grid with point number of $640\times160\times200\approx20M$ is used. For not very short waves, effect of ship motion may not be neglected in the added resistance prediction. Accurate calculation of the ship motion by the CFD method requires that the grid point is clustered around the whole ship, not just in the vicinity of the bow as used for the motion fixed case. Besides wave reflection and breaking at the bow, accurate simulation of interaction between the ship and incident waves is necessary in the computation. The Parallel version of the CFD code is used. Computations are performed in a PC cluster system with 64 nodes. The computation domain is divided into 16 sub-domains for parallel calculation.

Figure 3 shows comparison of RAOs between CFD and experiment. Good agreement is found for the heave motion. The computed pitch motion is lower than the experiment for $\lambda/L>1.0$. As a consequence, the predicted added resistance for $\lambda/L > 1.0$ does not compare well to the experiment. Insufficient computation domain size for long waves is considered as a reason for such discrepancy. In Fig.4, computed wave patterns for $\lambda/L = 0.6$ and $\lambda/L = 1.2$ are shown. The wave-ship interaction is well simulated by the present CFD code. For the case of long wave, wave reflection at the side walls of the numerical wave tank is clearly seen. Effect of such reflection wave on the added resistance calculation will be a research topic in the future. Nevertheless, the CFD prediction of added resistance in short waves is reasonably good for the present case.



(a) Heave (b) Pitch (c) Added resistance coefficient

Fig.3 Comparison of RAOs between experiment and CFD



(a) $\lambda/L = 0.6$ (b) $\lambda/L = 1.2$

Fig.4 Computed wave pattern for difference wave length.

References

[1] Faltinsen, O. M., Minsaask, J. and Liapisn, et al., Prediction of resistance and propulsion of a ship in a seaway. *Proceeding of 13th Symposium on Naval Hydrodynamics*, (Tokyo, Japan), 1980, pp. 505-529.

[2] Fujii, H., Takahashi, T., Experimental study on the resistance increase of a ship in regular oblique waves. *Proceeding of 14th International Towing Tank Conference*, 1975, pp. 351-360.

[3] Hu, C. and Kashiwagi, M., A CIP-Based Method for Numerical Simulations of Violent Free Surface Flows, *J. Marine Science and Technology*, 2004, V. 9, pp.143-157

[4] Yabe, T., Xiao, F. and Utsumi, T, The Constrained Interpolation Profile Method for Multiphase Analysis, *J. Comput. Phys.*, 2001, V.169, pp. 556-593

[5] Xiao, F, Ikebata, A., An Efficient Method for Capturing Free Boundaries in Multi-fluid simulations, *Int J Num Methods Fluids*, 2003, V. 42, pp. 187-210